

AddCAD 2014
for AutoCAD

User's Guide AddCAD

Tecnobit srl

Contents

Chapter 1	1 Getting started
	1 Consulting documentation
	1 AddCAD License Manager
	6 USB Activation
	6 Starting AddCAD
Chapter 2	8 News in AddCAD 2014
	8 New commands and functionalities
	8 Rectangular walls
	9 Polylines detracting
	10 Changing fixed points
	11 Commands and functionalities changes
	11 Assignment of render materials
	12 Dynamic updating for 3D walls
	13 Tapered pillars
	14 Changing walls face layer
	14 Genview and 3d lines
Chapter 3	15 Layout drawing
	15 Floor plan layout drawing
	15 Drawing walls
	19 Wall directly
	19 Rectangular walls
	21 Angle finishing, CLEAN, intersections
	22 Constructions with triangles
	23 Drawing stratifications
	24 Stratification options
	25 Multilines in walls
	27 Erasing walls
	27 Modify walls

Contents

	28	Join lines
	29	Planimetric relief
	31	Drawing audit
	32	Structural mesh
	34	Management of overlap states
	35	Overlap state generation options
Chapter 4	37	Parametric objects
	37	Introduction
	37	General considerations
	38	Parametric and Stair layer options
	39	Common features and commands
	39	Choice of an object
	40	Parameterizing objects
	42	Inserting objects
	44	Modifying objects
	46	Applying parameters
	47	Replacing objects
	49	Parameters sets
	50	Block and entity layers
	51	Changing layers to entities of an object
	51	Changing fixed points
	52	Viewing fixed points
	52	Different generations of objects
	54	Openings, doors and windows
	54	Insertion of openings
	57	Skylights
	58	Wall border, sloping parameters
	59	Wall break
	60	Layers of doors and windows
	61	Opening drawing options
	62	Delete openings
	63	Move opening dimensions
	63	Use of several scale factors
	64	Position and text styles of dimensions
	65	Lighting and plaster deduction

Contents

	66	Pillars in distribution layout
	67	Balconies
	68	Gates and 3d trees
	70	Roof elements
	72	Erasing roof elements
	72	Furnishing stairs
	75	Parametric stairs
	77	Sports facilities objects
	78	Objects and system for scaffolding
Chapter 5	80	Three-dimensional models
	80	Walls
	80	Introduction
	80	Constant height 3D walls
	81	Generation of an individual façade
	82	Variable height walls
	84	3D walls attached to slabs and stairs
	85	3D wall generation options
	87	3D wall settings
	87	Breaking lines and arcs
	88	Wall façade hatching
	89	Parapets
	90	Drawing eaves
	92	Generating railings and balustrades
	95	Creating 3D elevation structures
	96	Modifying 3D elevation structures
	98	Generating polygonal surfaces
	98	Edge visibility
	98	Hatch in 3D space
	99	Digital Terrain model
	99	Digital terrain model
	100	Elevation point
	101	Terrain model contours

Contents

	101	Calculating triangular model
	102	Discontinuity lines
	103	Viewing coplanar areas
Chapter 6	104	Creation of frames and holes in walls
	104	Introduction to frame definition
	105	Holes in walls
	105	Layout definition and frame labels
	108	Frame definition
Chapter 7	111	Roofs and floors
	111	Introduction
	111	Floor slabs
	114	Single roof slabs
	116	Examples of roof generation
	119	Hip roofs
	121	Gable roofs
	123	Barrel vaults
	125	Cross vaults
	126	Hatching roof slabs
	126	Roof girders
	128	Girder regeneration
	130	Roof editing
	130	Roof edit
	134	Intersection points on roof slabs
	134	Manage roof vertices
	135	Joining three roof slabs
	136	Isohypse lines
Chapter 8	138	Stairs
	138	Stair modeler
	139	Definition model
	139	Creating stairs from model
	143	Interactive definition of stair models

Contents

Chapter 9

- 144 Modifying stairs
- 145 Handrails
- 148 **Views**
- 148 2D plan and 3D views
- 148 Viewing roofs and floor slabs
- 149 Perspective views
- 150 Generation of views
- 150 3D views to line models
- 151 Line model with GENVIEW
- 155 GENVIEW with object selection
- 156 Updating views
- 156 Generating views of objects
- 157 Transferred shades
- 157 Shade generation options
- 159 Shade generation

Chapter 10

- 162 **Render POV-Ray and VRML**
- 162 Rendering options
- 164 Exporting scene in POV_Ray
- 166 Definition of customized material
- 168 Assigning materials to layers
- 169 Information for using POV-Ray
- 170 Inserting lights in drawing
- 172 Exporting in Vrml format
- 174 Assigning material for VRLM
- 175 Changing 3D wall face layers
- 175 Managing different layers on façades

Chapter 11

- 177 **Floors management**
- 177 Floor elevation
- 178 Floors management and visibility
- 180 Copy and translate floors
- 181 Multi-floor design

Contents

	183	Current floor
	183	Turn off floors
	184	Select a floor
Chapter 12	185	Printing
	185	Plot scale factors
	186	Sheet squaring
	186	Scale factor for layout viewport
	187	AddCAD drawing templates
Chapter 13	190	Dimensions
	190	Introduction
	190	Dimension options
	191	Room dimensions
	192	Internal wall dimensions
	193	Dimension of plans
	196	Dimensions for vertices
	197	Elevation views dimensions
	197	Multiline dimensions
	198	Dimension symbols
Chapter 14	199	Quantities survey system
	199	Survey system options
	200	Functional areas
	201	Under-roof functional lines
	201	Floor volume
	202	Deduction of lighting areas
	202	Ventilation surface parameter
	203	Room data calculation
	203	Introduction
	204	Room information
	204	Room data calculation
	206	Data for variable height room
	208	Polyines detracton

Contents

	209	Highlighting room with fixed areas
	210	Tab total
	210	Room data updating
	210	Number of rows and tab name
	211	Modifying descriptive room data
	211	Reassigning room numbers
	212	Generating polylines of rooms
	213	Room data extraction
	213	Tabs and labels
	213	Tabs and labels
	215	Room label with tab
	226	Room label without tab
	228	Opening tabs
	228	Opening indexing system
	229	Inserting and updating opening tabs
Chapter 15	233	Utility commands
	233	AddCAD license and version
	233	Block catalog
	233	User block library
	235	Add and modify a block definition
	236	Layer utility
	236	Empty all entities from the layer
	237	Layer activation commands
	237	Visualize only selected layers
	237	Floor coloring and dithering
	238	Cursor rotation
	238	Rotate with a second reference
	239	Move in Z direction
	239	Offset with layer change
	240	Line rectangles
	240	Breaking entities
	240	Properties of entities
	241	Multiple distances
	242	Numbered labels

Contents

Chapter 16	244 Options and configurations
	244 Layer generation attributes
	245 Names for wall layers
	246 Searching definition files
	246 AddCAD options
	247 Definition of new stratification
Chapter 17	250 AddCAD toolbars

Consulting documentation

There are several functions which make it easy to learn and consult the various topics of the program. We recommend reading the various topics in the summary and practicing at the same time.

This is the only way to learn all the important functions of AddCAD.

Links to pages of related topics

- When you find blue underlined words in the text, pressing this text with your mouse will open the page to explain the concerned topic.

Toolbar pan

- [The toolbar page](#) is made in such a way that each button is linked to the associated function. So to know what a certain button is for, just go onto it and press the left mouse button.

Information contained in page title

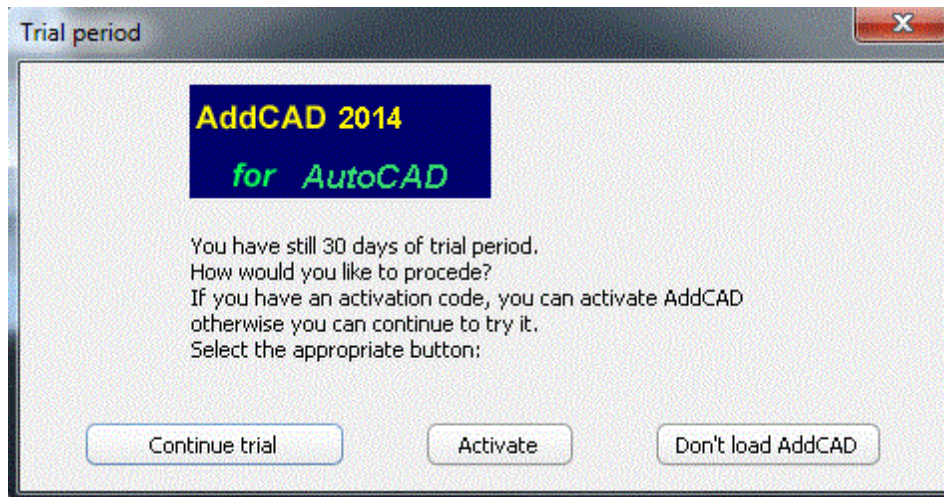
- When you read a page and a command is associated to the topic, you can see which button on the toolbar starts it up. Just move your mouse onto the page title and press the left key.

AddCAD License Manager

In order to activate the AddCAD software, you need a *License Serial Number*. You can obtain the *License Serial Number* from your AddCAD reseller. AddCAD can be activated directly on your computer or in a normal USB pen drive. To discover the advantages of the USB method, please see the [USB Activation](#) page.

Running the trial version

When you start AddCAD, if it has not yet been activated, you will see the following dialog box:

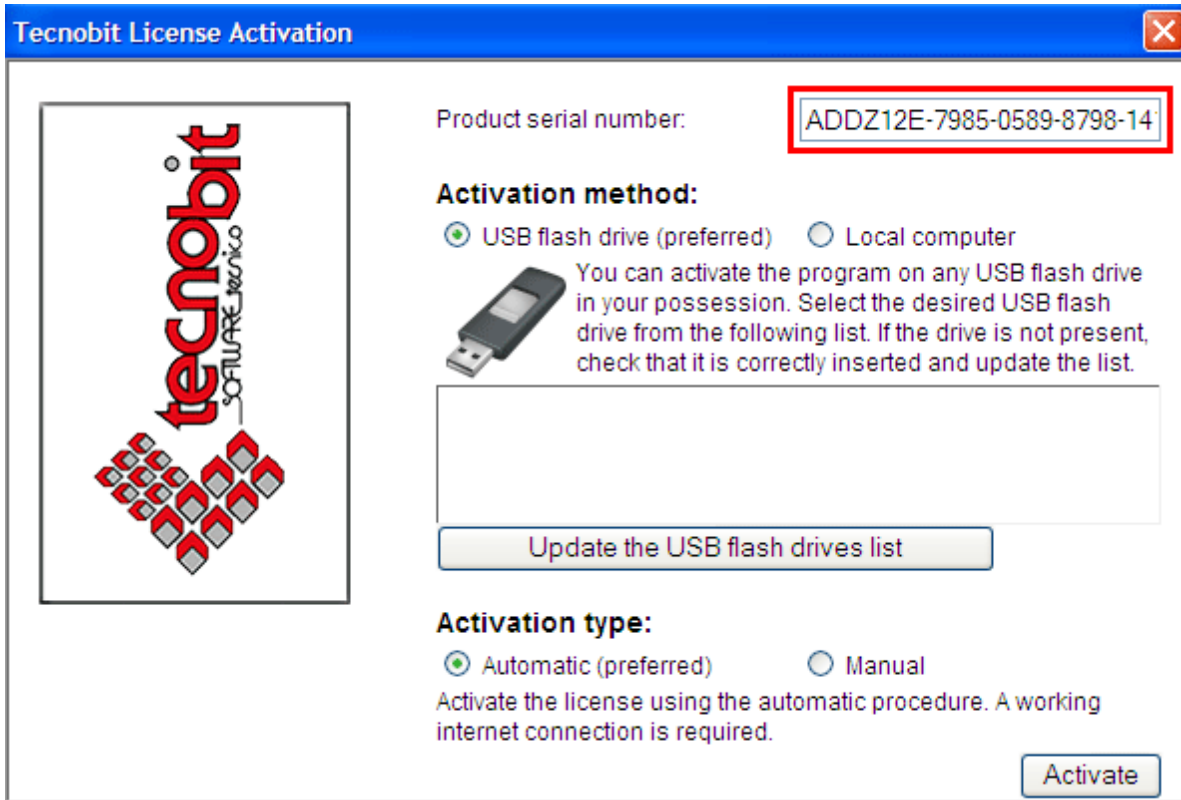


If you haven't purchased the license, you can use the trial version by clicking the *Continue trial* button. As explained in the dialog box, this version allows you to fully use all AddCAD features without any restriction.

Activating the program

If you have purchased the software license, you need to activate it performing the operations described below. In order to obtain an immediate activation, you must have an internet connection in the computer where you have installed AddCAD. Otherwise, the activation can be obtained via e-mail. Both methods are described here below.

Click the *Activate* button in the dialog box shown in the [Running the trial version](#) paragraph above; the following window will appear. In the cell at the top, insert the serial number you obtained from your AddCAD reseller (via e-mail or inside the package of the product).



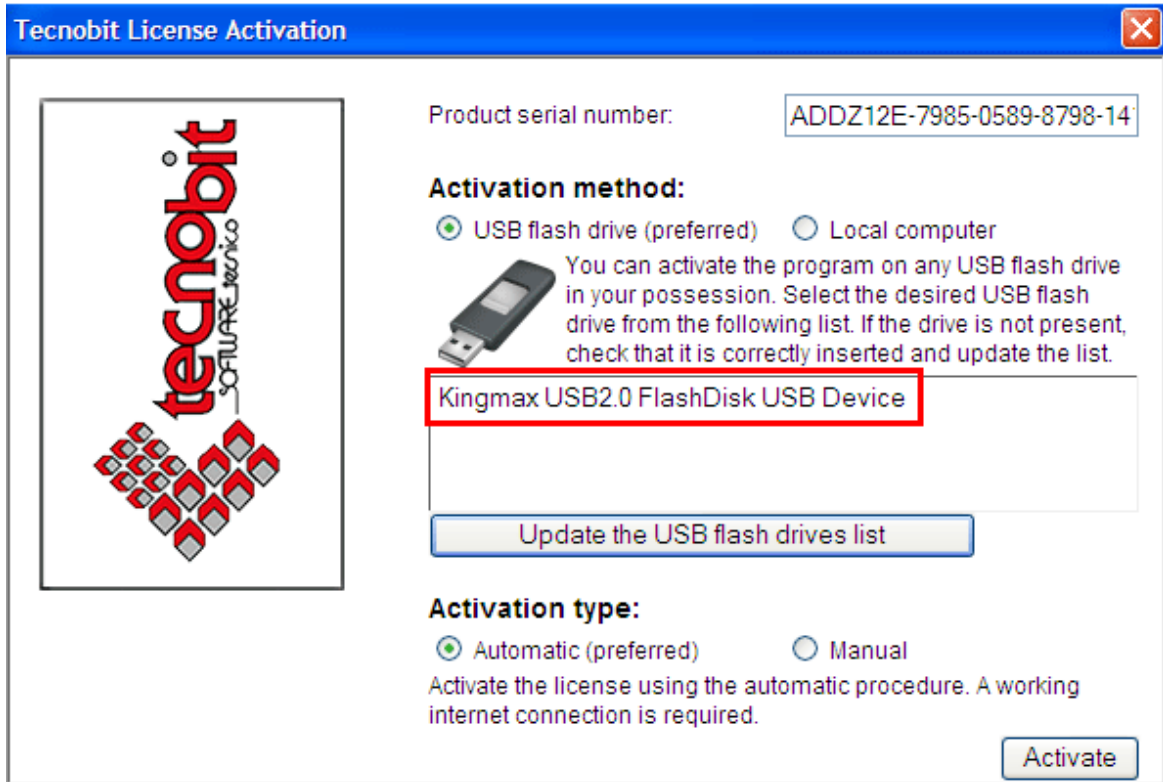
Note: the serial number shown in the window above is of a mere explanation purpose only and is not valid for activation. The actual serial number to be entered is the one you receive when you purchase the software.

Once you have entered the serial number, you have to select the options for the activation method and type.

Activation method

This option allows you to choose whether to activate the software on the computer or on an USB pen drive that you already own, i.e. you don't receive any dongle from your AddCAD reseller, you just use a normal USB pen drive of your own (please see the [USB Activation](#) page for more information).

- **Local computer** (not recommended): choose this option to activate the license on the computer where you AddCAD is installed. Please note that, currently, by using this method you cannot transfer the license from one computer to another, even though this option will be available in a future AddCAD upgrade. Therefore, if you want to use the software on multiple computers (but not simultaneously), even though you have purchased a single license, it is recommended to choose the USB activation as explained below.
- **USB flash drive** (recommended): for this method you need to provide a normal USB pen stick which will become the license key to run the program. To check if a USB pen drive is suitable for this purpose, just plug it into one of your computer USB ports and press the *Update the USB flash drives list* button. If the device name appears in the box, as in the figure below, it means that it has been recognized and can be used.



So, just select the USB device name in the box. Please note that in case you have more than one USB stick connected to your computer, the box will contain the name of all of them and you then have to select the desired one.

Activation type

This option allows you to choose whether to activate the software using an automatic procedure via the Internet or a manual operation via e-mail.

- **Automatic** (preferred): select this option only if the computer where AddCAD is installed has Internet access, then click the *Activate* button and a message will appear informing you that activation is successful and the program will start. In case of Internet connection problems, please carefully read the message window in order to see the possible cause and remedy, if you are not able to solve the problem, please choose the *Manual* method explained here below.
- **Manual**: select this option if your Internet access is available on another computer, not on that where AddCAD is installed. Once you have selected it, the activation window will change as following:

TecnoBit License Activation

Product serial number:

Activation method:

☒ USB flash drive (preferred) ☐ Local computer

You can activate the program on any USB flash drive in your possession. Select the desired USB flash drive from the following list. If the drive is not present, check that it is correctly inserted and update the list.

Activation type:

☐ Automatic (preferred) ☒ Manual

Send an email containing the serial number (above) and the following installation code to activation@addarcsoft.com

Paste here the license code you'll obtain from activation@addarcsoft.com

As explained in the window, all you need to do is to send an email to activation@addarcsoft.com including:

- your product serial number;
- your installation code (indicated in red in the window here above).

Once TecnoBit customer care has received your email, you will be sent your license code. This code is composed of 3 lines of text like the following (the license code here below is of mere explanation purpose only and is not valid for activation):

```
LICENSE tecnobit addcad 2.01 permanent uncounted hostid=789eeb30
_ck=640dfc5632
sig="60PG4580JJG64EJ20AWHUGAK30QT5448968BN3Y108A1YVB
KRWU2KPDKTKYJX5VUXH8DE30DD2A0"
```

Paste the activation code you have received in a text file and go to the computer where AddCAD is installed. Copy the code from the text file again and paste it into the box at the bottom of the activation window. Finally click the *Activate* button. If all the above described operations have been performed correctly, a message will appear informing you that activation is successful and the

program will start. Otherwise, if you get an error message, it means that there were some errors while communicating the serial number and the installation code or while copying/pasting the license code. In this case, please repeat all the procedure paying attention to all character of the strings.

USB Activation

AddCAD adopts a new technology that allows you to activate the license in a normal USB you already own. This means that you don't receive any dongle from your AddCAD reseller but you just use a normal USB pen drive of your own that you used to store your files. The activation is performed via the product serial number provided by your AddCAD reseller when you purchased the product. Thereafter, whenever you want to use the program, all you need to do is to insert the USB pen drive in your computer. This memory stick can still be used as a normal data storage but becomes your license key for AddCAD. Therefore, if you run the program without inserting this pen drive, a warning message will appear and AddCAD will not run.

Using the USB key, you can then use AddCAD on multiple computers, even though not simultaneously. This means that, once you have activated AddCAD pen drive on the first computer, you can install the program also in other computers and use it by simply inserting the pen drive on each of them.

For this reason, the USB activation is very useful in case you want to use AddCAD on both a desktop computer and a laptop.

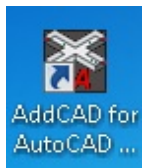
The activation in fact resides in the USB pen drive and is independent of the computer, as such, therefore, it avoids the risk of losing the activation in case of disk formatting or computer replacement.

The USB activation does not imply additional cost to the user as Tecnobit does not send him any dedicated hardware key (dongle) but, as mentioned, the user simply use one of his own USB sticks (or buys one) to activate the software. You can also activate multiple Tecnobit programs on the same memory stick (one license per product). It is therefore advised to attach a label to the pen drive showing the name of the software activated on it.

Important Note on USB stick

Once the program has been activated, it requires that the USB pen drive is always inserted in one of the computer USB ports. The loss of the memory stick will also result in the loss of the program license. It is therefore recommended to treat the device with care and keep it in a safe place.

Starting AddCAD



During installation, a new profile is created inside AutoCAD Options. This profile contains all the variables and settings to operate AddCAD. The profile can be modified to better meet your personal needs. By starting AddCAD from its icon on the desktop, the profile named AddCADINT is chosen directly.

Note for AutoCAD LT users

AutoCAD LT does not have the capability to manage multiple profiles. Therefore, during installation, the unique AutoCAD LT profile is modified so that AddCAD can work properly. For this reason, when you install AddCAD on AutoCAD LT the new link icon in your Windows desktop is not created. So, to start AddCAD for AutoCAD LT you simply need to double-click the AutoCAD LT link icon that you already had before installing AddCAD.

Drawing models and setting preferred units of measurement

AddCAD can work starting with any drawing model or opening any existing drawing. Make sure you are working with the right measurement units before starting to draw. If you begin a new drawing starting off from one of the models supplied with the program, the measurement unit is chosen by choosing the specific model. AddCAD proposes the following [drawing templates](#).

Addcadmm.dwt	set up to work in millimeters with plot styles depending on the color (CTB)
Addcadcm.dwt	set up to work in centimeters with plot styles depending on the color (CTB)
Addcadmt.dwt	set up to work in meters with plot styles depending on the color (CTB)

The drawing models can also be custom made to adapt them to personal needs. For example, new layout configurations can be modified or defined to speed up the design plotting stage.

Work with AddCAD on a drawing started only with AutoCAD

If you work with a drawing started with a model not supplied with AddCAD, the measurement units will be in centimeters. This could cause some problems if the drawing was made in meters. Use the command ADDOPTIONS to change the unit of measurement. Enter the correct unit of measurement in Parametric and stairs layer tab and press Apply.

An even better solution would be to select all the entities of the old drawing (activate all the layers before selecting), to copy them with COPYCLIP, to open a new drawing starting from the specific model and to use PASTECLIP to paste all the entities in it.

Note on UCS and AddCAD

Many AddCAD commands assume that the UCS taken by the user is the global one. Should any problems occur, make sure the UCS is the global one.

New commands and functionalities

Rectangular walls

With this command AddCAD 2014 introduces the concept of a link between the two lines of a wall. In practice the four lines generated by this command have a unique code that groups them together. This link allows to solve many problems of recognition of the walls for both the stratification drawing in the plan view and for the closure on the top. Let's begin by examining in detail the command options, then we will see how some situations are completely resolved.

Command: RWALL

Specify the first endpoint of the diagonal or [Layer]: <Option Layer or first point>

Specify the second endpoint of the diagonal or [Thickness]: <Option Thickness or second point>

If you choose the *Layer* option, you can change the current layer to draw the wall. In this case, once you have selected the new layer, the program asks you to specify the first point. The following result depends on the action you take in response to the second question. If you specify the second vertex, it is taken as the second point of a rectangle diagonal. The rectangle will then have its sides parallel to the coordinate X/Y axes. If you choose *Thickness*, because it is a known value of the project, then the subsequent questions are as follows.

Specify the second endpoint of the diagonal or [Thickness]: T

Enter the thickness of the rectangle wall <0.300>: <Specify the new thickness or Enter>

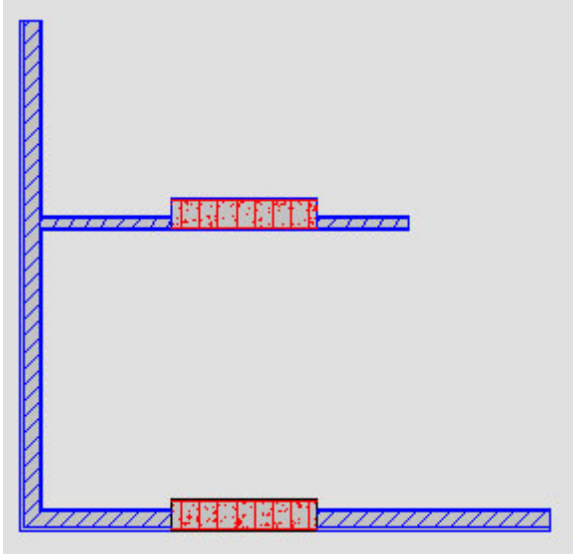
Second endpoint of the rectangle base: <second point>

In this case, once you have assigned the thickness, you can freely position the second point wherever you want, assuming that it is the second end of the triangle base this time. In this way you can rotate the rectangular wall as you desire.

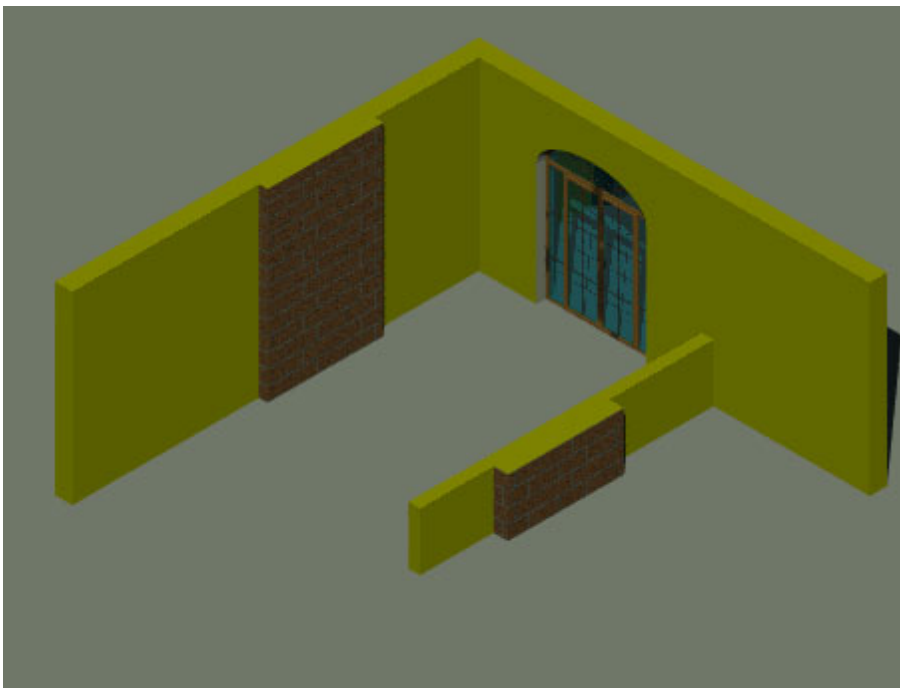
The command is cyclic, it continues asking points for the next rectangle. To exit the command simply press Enter.

Please note that, if at a later time you erase one or both of the side closure segments of the wall, or the lines are modified by stretching their vertices, the automatic recognition of the wall will still take place.

However, remember that, in order to obtain the recognition of the wall, the *Max thickness for the walls* parameter inserted in the [Openings and walls 2D/3D Tab](#) of AddCAD preferences must be greater than the thickness of the rectangle wall that you draw.



In the case of walls of different materials and thicknesses, the [recognition and the drawing of the stratification](#) occurs without errors. In addition, as you can see, you no longer need to insert a separator.



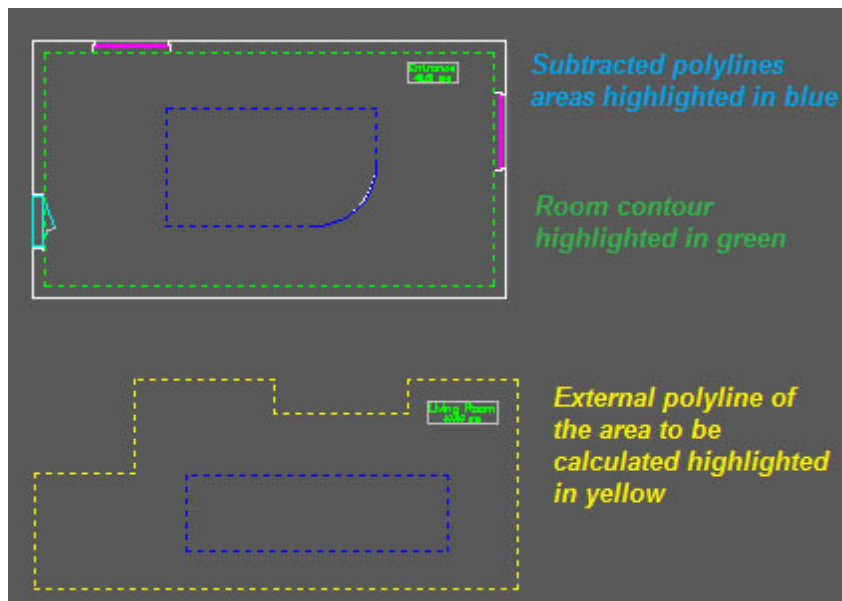
The creation of 3d walls with [top closure](#) occurs without errors and thanks to the new recognition algorithm you can obtain the closure for walls that have different heights.

Polylines detracton

You can defined, by means of a closed polyline, any area to be deducted from the quantity calculation inside a room or inside a polyline of an area to calculate. The polylines to be deducted are automatically recognized and their contour is highlighted with blue color just like the program does for [stairs and pillars](#).

The POLYDEDUCT command allows you to assign and remove the feature of

deductibility of polylines. Obviously the condition for the correct deduction is that the areas bounded by the polylines are internal to the room area or to the polyline of an external contour.



Command:
POLYDEDUCT

Select a closed polyline to which you want add deductibility[Remove deductibility]/Enter to finish:
<Select>

Select a closed polyline to which you want add deductibility[Remove deductibility]/Enter to finish:
<Enter>

As you can see, you can also remove the deductibility feature of polylines. By choosing the *Remove* option, the command prompts you to select the polylines to which you want to remove the deductibility.

Changing fixed points

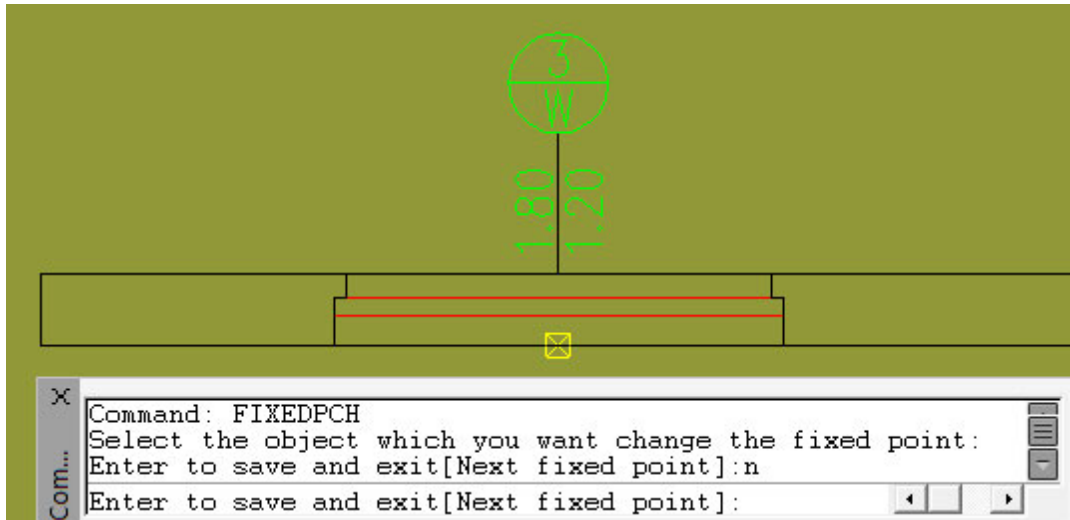
The [fixed points](#) of parametric objects are important because they represent the points that do not change their position in the drawing after a modification of the object. An object fixed point can be selected when you insert or edit the object. In AddCAD 2012 version, we introduced the capability to [display the objects current fixed points](#). In this version you can also change the current fixed point at any time using the new command FIXEDPCH. This command, once it has changed the fixed point to the selected object, allows you to select other similar objects to which you want to change the fixed point too.

Command: **fixedpch**

Select the object which you want change the fixed point :

Enter to save and exit[Next fixed point]:<Enter or N>

Select other objects of the same type, to which you want assign the same fixed point: <select objects of the same type>



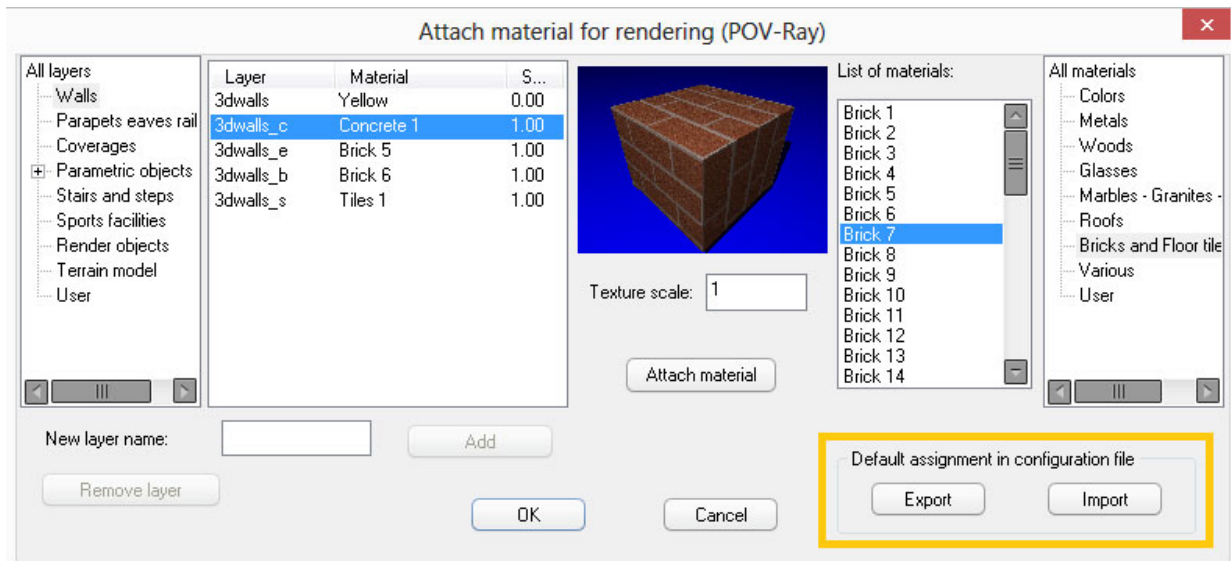
Commands and functionalities changes

Assignment of render materials

In this version the [POVMAT command](#) stores the material assignments on the drawing basis and not longer generically in the external configuration file. In the previous versions there was the following drawback:

- 1) in a project you assign a certain material to a layer;
- 2) then you start a new project and in this one you make different assignments;
- 3) when you perform a new rendering of the first project you “surprisingly” obtain the assignments of the second project.

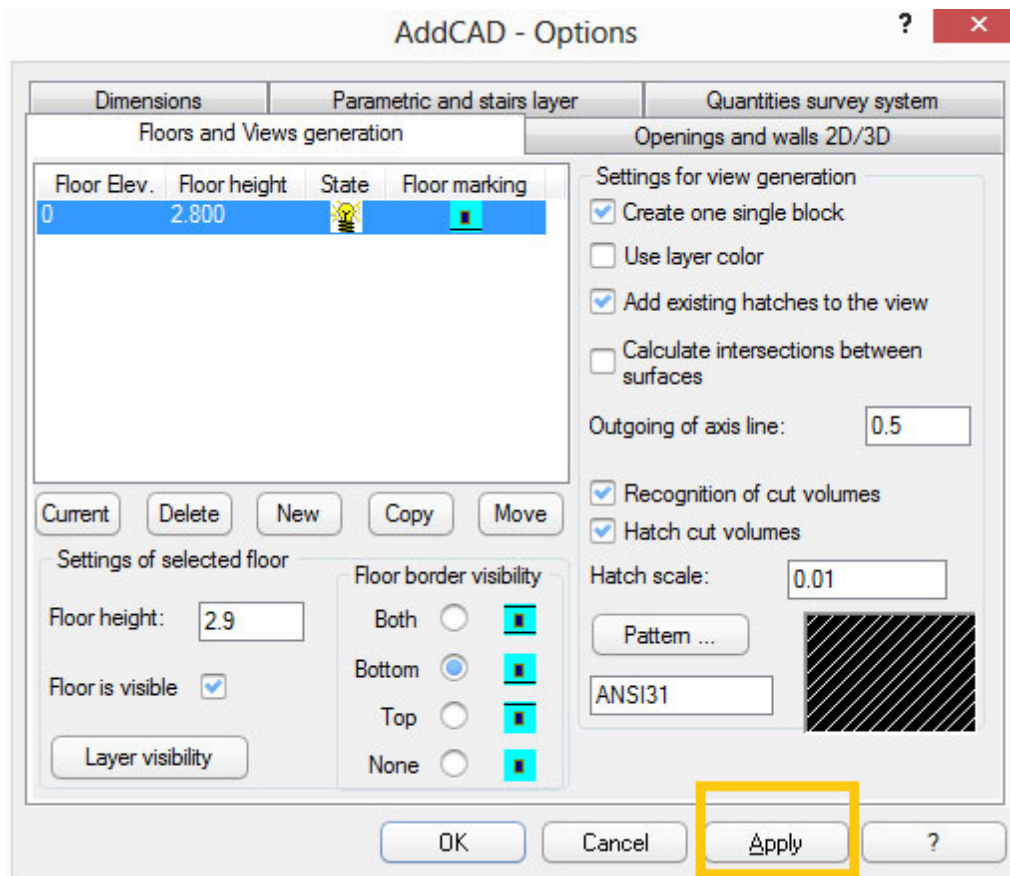
Now assignments are stored directly in the DWG file in an appropriate internal dictionary. As you can see in the figure below, however, the new command still allows you to change the default configuration by saving in the file the associations made in the drawing and alternatively to reassign the stored associations back in the drawing, thus recovering a standard situation. When you start a new drawing the assignments material- layer are those of the standard file.



The new dialog box for assigning materials to layers includes two new buttons through which you can export in the base configuration file the preferences of assignments made in the current drawing, and conversely import the configuration file settings into the current drawing.

Dynamic updating for 3D walls

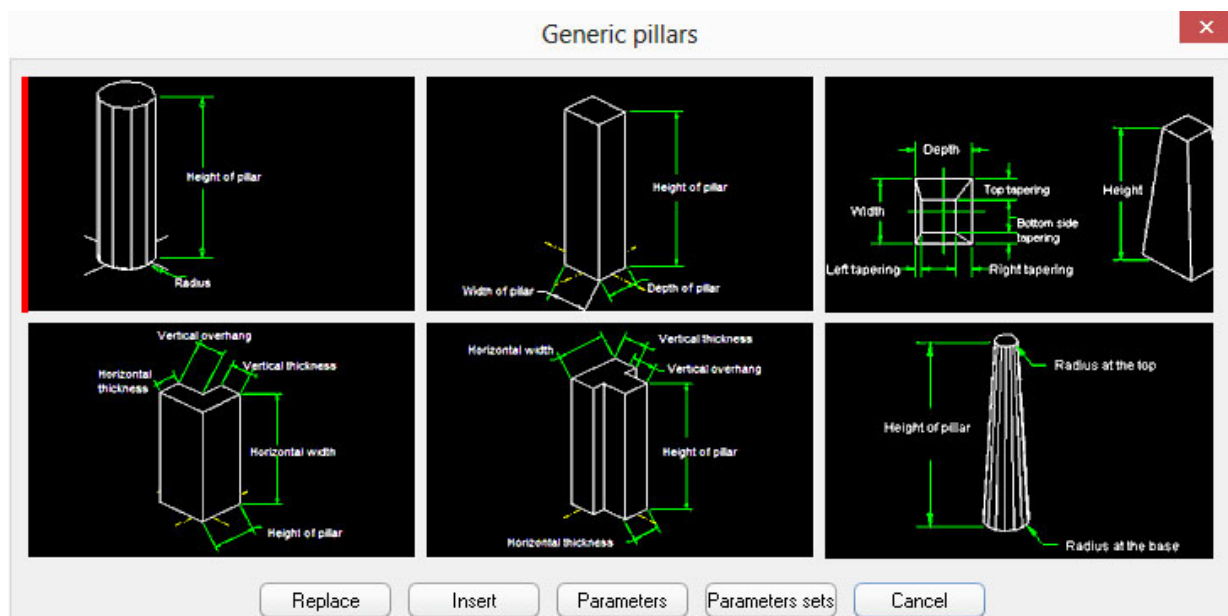
In the [Floors and views generation tab](#) you can directly regenerate the model of 3D walls accordingly to the changes to the visibility of the upper/lower edges (floor marking). If you change the visibility state or the floor height, the *Apply* button is automatically enabled and, if you click it, you get the regeneration of the walls for all floors to which you have changed the visibility status.



The **Apply** button is enabled if you change the *Floor height* or the *Floor border visibility*. In both cases you can update the drawing contents by simply clicking it.

Tapered pillars

In the window for selecting generic pillars, two new pillars have been added, one circular and the other rectangular, together with the capability to parameterize their different section size at the base and at the top, so you can get tapered pillars.



The new window for generic pillars: on the left the two new types of pillars.

Changing walls face layer

Both commands *Change 3D wall layers (XLW3D)* and *Change layer wall 2D (Wallml)* allow you to iteratively select the lines of the walls. This capability speeds up the layer change for the walls. At the end of the selections you can easily stop the command by simply pressing Enter.

Command: *xlw3d*

Select a line or arc of a facade or [Outside]:<Select>

.....

Select a line or arc of a facade or [Outside]:<Enter>

Command: *wallml*

Select a line or arc of a facade or [Outside]:<Select>

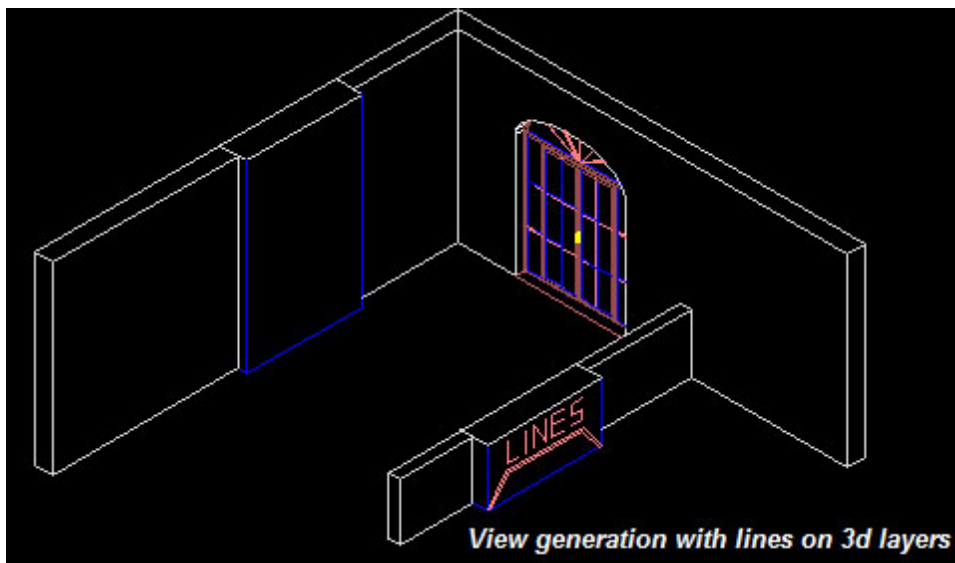
....

Select a line or arc of a facade or [Outside]:<Enter>

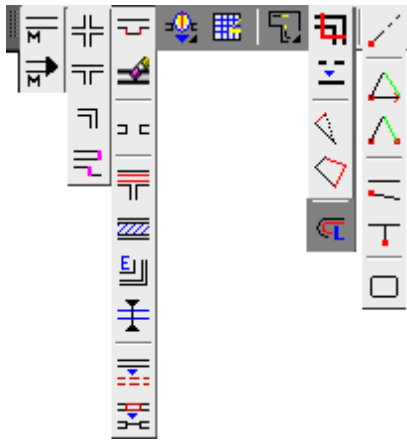
The XLW3D and Wallml commands perform the selection in a cyclic way.

Genview and 3d lines

In this version the *Viewgeneration (GenView)* command also considers the presence of lines on 3D layers. In practice, it processes all the line segments present on layers that have names starting with '3D' or with 'ADDC3D'.



Floor plan layout drawing



The plan distribution drawing of walls is performed with the WALL and WALLD commands.

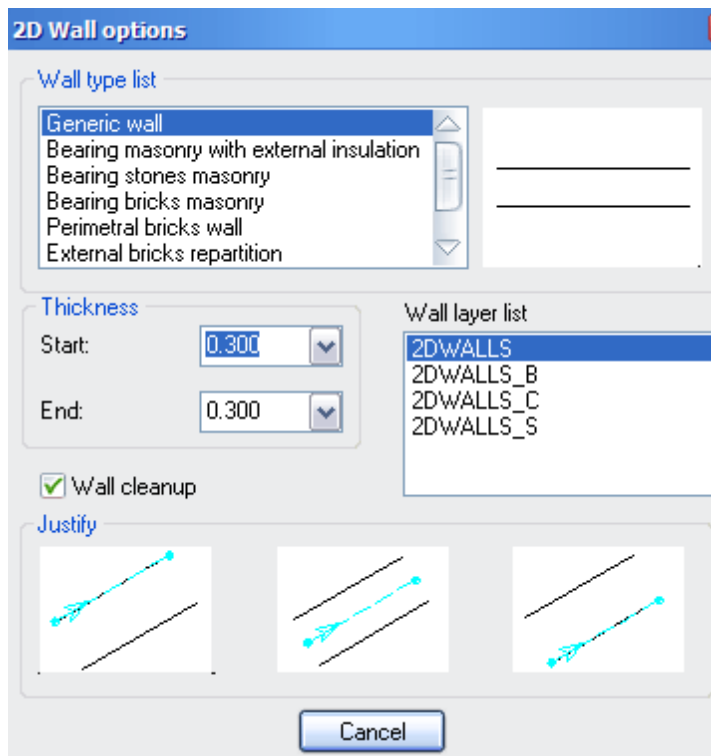
The entities representing 2D walls are normal AutoCAD lines and arcs. This means that AutoCAD commands can also be used to draw walls. As we will see, the condition for AddCAD to recognize lines and arcs as walls is simply the fact that these entities belong to certain layers. The commands which draw the walls automatically draw the lines, inserting them in specific layers and placing them at the appropriate Z elevation. Both of these commands have different functions which speed up the plan drawing, such as: the breaking and automatic selection of lines, automatic merging with arcs and modification of drawing options

while using the command. As walls are being drawn, the lines can automatically be attributed a wall type. Assigning this type has no immediate graphical effect, but it allows you to automatically calculate the wall and to draw it with its layered representation. A series of helpful commands makes it possible to simply and immediately intervene on the existing distribution drawing. Parametric openings and pillars can be placed in the drawing. These openings and pillars can be canceled by reconstructing the wall drawing the way it was before their insertion. Even entire wall segments can be canceled with one command, properly restoring attachments. It is possible to modify walls by changing their thickness and inserting niches and overhangs, even with openings. There are also three commands, CLEANL, CLEANL and CLEANX, to clean up joints. When the distribution needs to be modified, we recommend using the AddCAD commands as they are faster and less likely to make mistakes. Nonetheless all the AutoCAD commands can be used to process the distribution layout.

Planimetric relief

A series of commands allows you to draw the shape and size of rooms based on measurements taken on an existing building. Parallelism and perpendicularity constraints can be assigned and the layout rectified, with the usual triangulation methods, by means of relief measurements. The RECTANGL command allows you to quickly draw the shape of a four-sided room.

Drawing walls



The WALL command helps to draw the double lines and double arcs which represent the walls. Before starting to draw, the program requests that some choices be made in its dialog box.

It is possible to select or write in the thicknesses, the type of wall, the name of the layer to enter the entities and whether to automatically cut the lines. The wall justification closes the dialog box and has you continue to process the command.

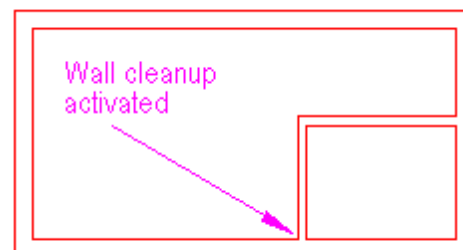
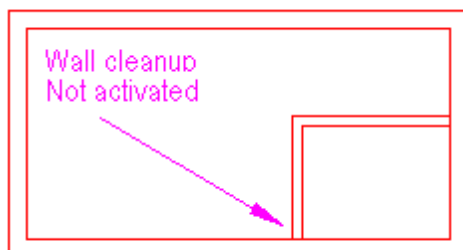
Wall type list

The list at the top contains the descriptions of the types of walls. An image is shown to the side representing the type of wall selected in the list. There is a command to [define and modify wall types](#) and one which allows you to

[automatically draw the layered representation](#) of the walls. Notice the presence of a generic wall. This is a default type of wall without layers. Usually walls are drawn without assigning them any type right away, an operation which can be done later on.

Wall layer list

The list on the right contains the names of the layers for the walls. Actually the list does not carry the complete name of the layer but normally just the root of the name. To correctly manage the mechanism with several layers, AddCAD links the element in the list with a particular prefix and suffix to obtain the layer name. There is the possibility of [adding new elements to the list](#). The choice of a type of wall can automatically select an element in the layer list matched to it. However the user can change this selection. The availability of several layers for walls makes it possible to give various walls different aspects, through the attributes of visibility, color, type and line thickness.

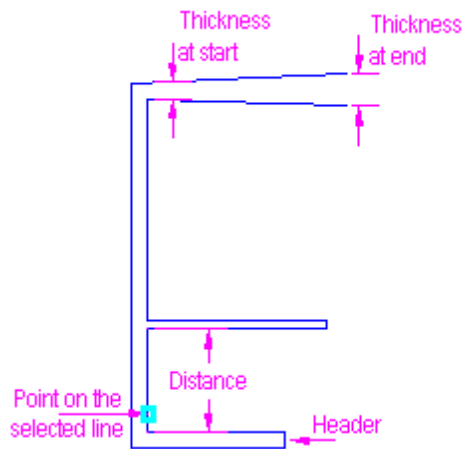


Thickness

It is possible to select the thickness from the list or to assign new and different thicknesses at the start and end of the wall. This is useful when you wish to draw walls with variable thickness. If the start thickness is modified without modifying the end thickness, the end thickness is automatically assigned the starting value.

Points are indicated while drawing the walls. These points can be in the middle of the double line (central) or on one of the two sides (right or left). Once the type of point alignment has been chosen, the program draws directly on the AutoCAD editor.

[Header/<Select>]First point:



Header

If the Header option is chosen, the command closes the end of the wall.

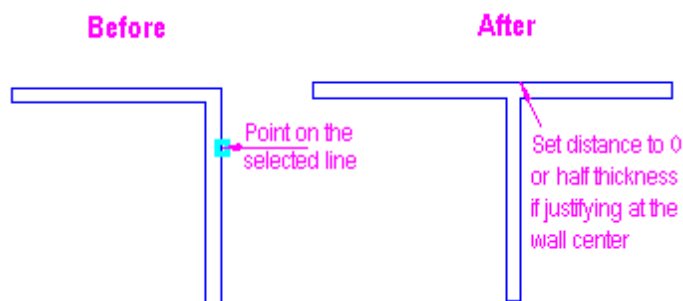
Selection or Enter

By choosing this option you are requested to select the wall line starting point. It is possible to measure the distance from the closest endpoint to the selection point of the line and to decide where to begin drawing the wall. In fact the command, positioning the origin of an elastic line at the endpoint closest to the line selected, requests the distance from the vertex to the attachment. The distance can be entered numerically or by indicating a point.

If a point is typed in close to the line of another wall of the same floor, the program automatically breaks the underlying line and connects the wall directly to the line.

If there are two lines belonging to the layout in the selection crosshair, like in angles, the command is not capable of identifying the line from which to start and the operation could have an unwanted result. When you wish to start from an angle, it is recommended to follow the solution shown in the figure to the right.

In succession, the command generates a set of walls with the correct finishing angles. During the drawing stage, besides the next point, the command presents a series of options:



[Undo/Arc/Close/Options/Select/ Header/Triangle]Next point:

As for all AddCAD commands, the options can be chosen from the context menu by right-clicking the mouse or by typing the capitalised letter from the options list.

A point or a numerical value

The command generates double lines of the last segment of the wall. If a

point is typed near another wall of the current floor, the command automatically generates the attachment and ends execution. If you wish to draw a wall of a certain length in the direction of the elastic line, just write the length.

Enter

The command draws the last section of the wall and ends execution.

Header

The command draws the final header of the wall segment just drawn and ends execution.

Close

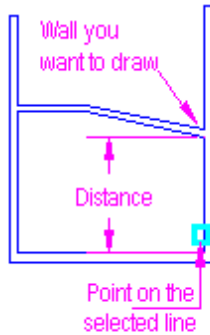
The command links the current wall segment to the start of the first wall and ends execution. This option is not available if you start with the Header option.

Undo

This option allows you to undo a point at a time as far as required to the start of the command. We therefore recommend using it if incorrect points are entered. Finally pressing **Esc** cancels all the operations performed by the command.

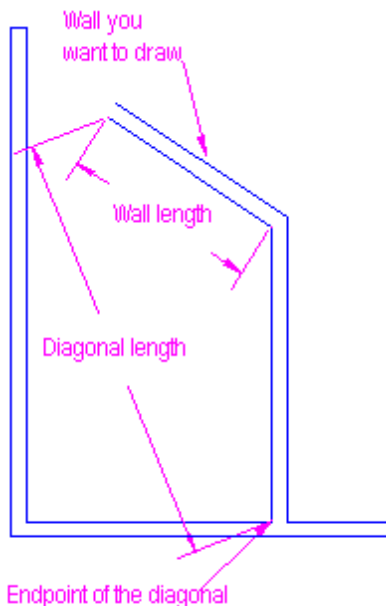
Options

The command is suspended temporarily to display the dialog box and to allow you to change the type of alignment, thicknesses, layer and wall type. When the type of alignment is chosen, the WALL command resumes drawing the lines with the new set values.



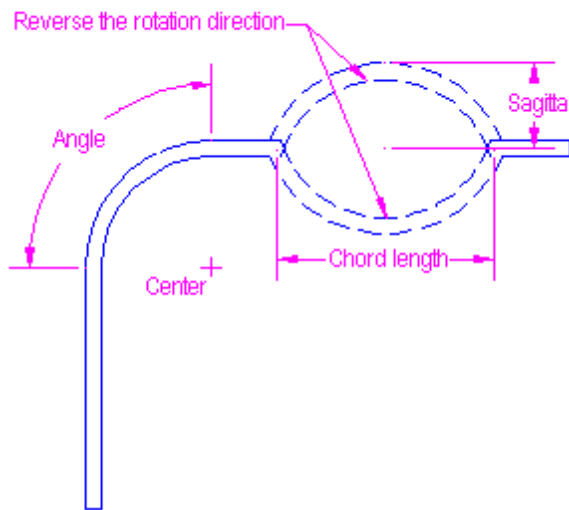
Select

This is for linking the wall being drawn to another existing wall, at a certain distance from the endpoint closest to the point selected. If this option is chosen, AddCAD requests to select a line of the wall. Once the wall has been selected, it attaches an elastic line to the endpoint closest to the line selected and requests its distance.



Triangle

The Triangle option solves all those problems in which two measurements are known which determine the length and position of a wall segment. When the Triangle option is activated, the command requests the data which determine the new wall segment. The information entered concerning wall length, diagonal length and the point from where it should be measured can give two graphical solutions. By moving the cursor from one part of the previously drawn wall to another, you can pass between the two possible solutions.



Arc

The Arc option allows you to draw a curved wall with an arc section. You cannot draw two consecutive curved walls. A flat wall, even very small, must be placed between the two curved walls. The curved wall can be drawn in two different ways: by indicating the chord and then the sagitta, or by indicating the center of the arc and then the angle described by the wall. With the first method, the command requests:

[Angle/Reverse the rotation direction]Chord length:

The Angle option allows you to pass to the second method.

Reverse the rotation direction is an option which is always available while drawing the curved wall and allows you to rotate the arc in the direction opposite that displayed.

[Reverse the rotation direction]Sagitta:

The value entered is intended as the sagitta of the arc.

[Reverse the rotation direction]Rotation angle:

This last request allows you to rotate the curved wall respect to the wall drawn with lines in the previous section.

The second method requires the following data.

[Reverse the rotation direction]Center:

[Reverse the rotation direction]Angle:

Wall directly

The WALLD command has the same function as [the WALL command](#). The only difference is that it begins by directly requesting the initial point without passing through the settings dialog box. When you wish to draw more walls with the same settings, it is much faster to use WALLD because it avoids passing through the dialog box.

Rectangular walls

With this command AddCAD 2014 introduces the concept of a link between the two lines of a wall. In practice the four lines generated by this command have a unique code that groups them together. This link allows to solve many problems of recognition of the walls for both the stratification drawing in the plan view and for the closure on the top. Let's

begin by examining in detail the command options, then we will see how some situations are completely resolved.

Command: RWALL

Specify the first endpoint of the diagonal or [Layer]: <Option Layer or first point>

Specify the second endpoint of the diagonal or [Thickness]: <Option Thickness or second point>

If you choose the *Layer* option, you can change the current layer to draw the wall. In this case, once you have selected the new layer, the program asks you to specify the first point. The following result depends on the action you take in response to the second question. If you specify the second vertex, it is taken as the second point of a rectangle diagonal. The rectangle will then have its sides parallel to the coordinate X/Y axes. If you choose *Thickness*, because it is a known value of the project, then the subsequent questions are as follows.

Specify the second endpoint of the diagonal or [Thickness]: T

Enter the thickness of the rectangle wall <0.300>: <Specify the new thickness or Enter>

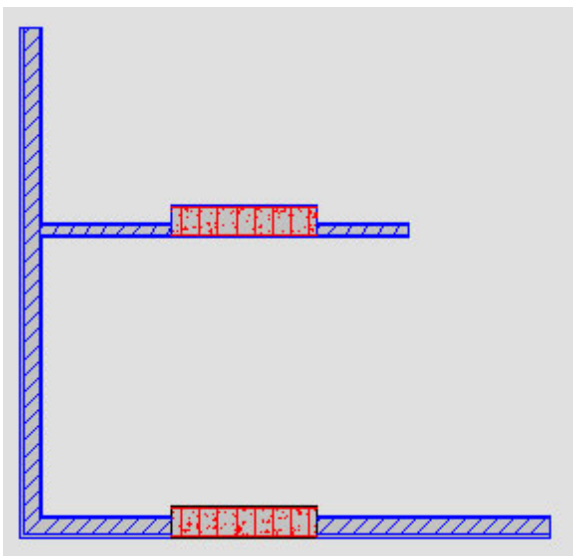
Second endpoint of the rectangle base: <second point>

In this case, once you have assigned the thickness, you can freely position the second point wherever you want, assuming that it is the second end of the triangle base this time. In this way you can rotate the rectangular wall as you desire.

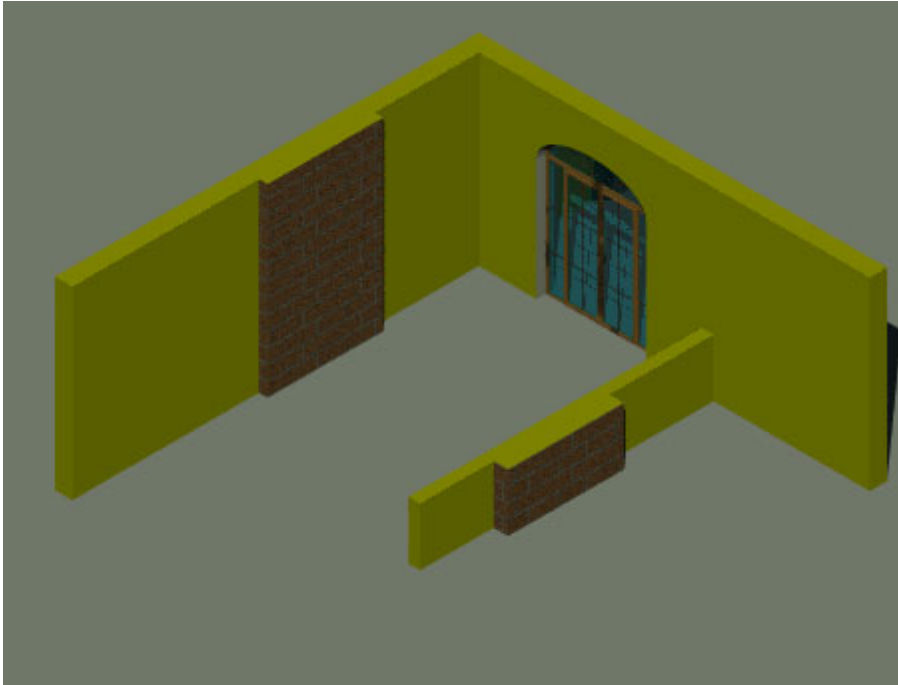
The command is cyclic, it continues asking points for the next rectangle. To exit the command simply press Enter.

Please note that, if at a later time you erase one or both of the side closure segments of the wall, or the lines are modified by stretching their vertices, the automatic recognition of the wall will still take place.

However, remember that, in order to obtain the recognition of the wall, the *Max thickness for the walls* parameter inserted in the [Openings and walls 2D/3D Tab](#) of AddCAD preferences must be greater than the thickness of the rectangle wall that you draw.

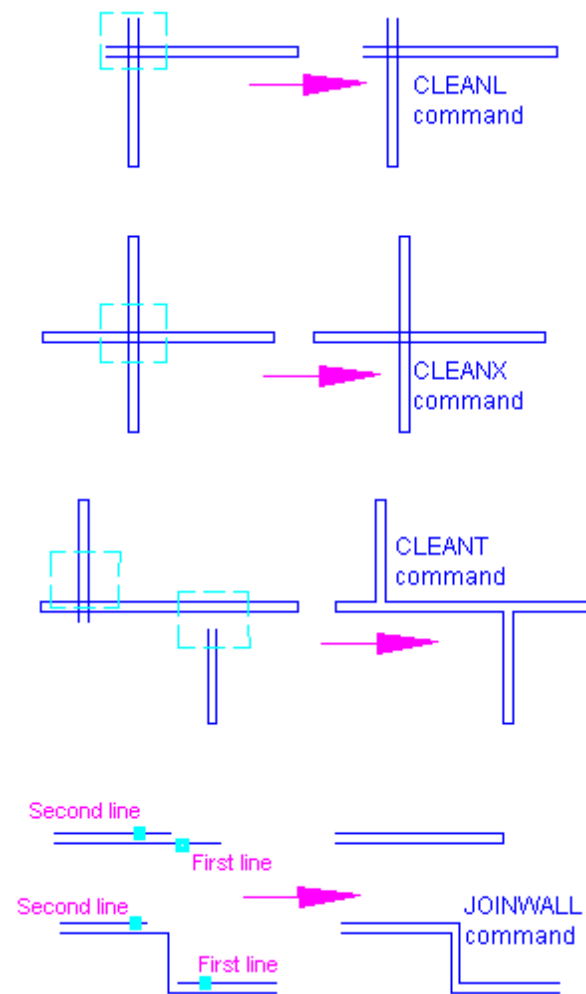


In the case of walls of different materials and thicknesses, the [recognition and the drawing of the stratification](#) occurs without errors. In addition, as you can see, you no longer need to insert a separator.



The creation of 3d walls with [top closure](#) occurs without errors and thanks to the new recognition algorithm you can obtain the closure for walls that have different heights.

Angle finishing, CLEANT, intersections



These are four commands which help to speed up finishing operations of the 2D plan wall drawing.

Angle finishing

The CLEANL command allows you to join the two pairs of angle lines with just one command.

Cross merging

The CLEANX command breaks lines at wall intersections in order to finish the intersection between four walls.

CLEANT merging

The CLEANT command breaks and joins lines at a junction such as wall CLEANT, in order to finish the union between the two walls.

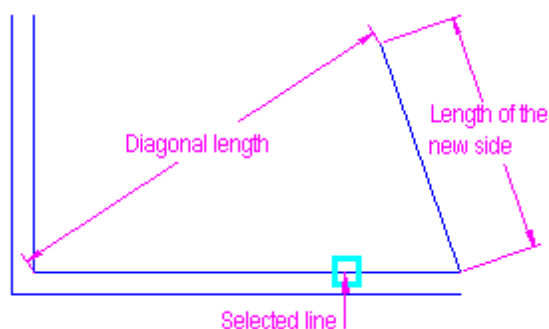
These three commands require you to select four lines belonging to the desired finishing. With the CLEANT command you can select three lines. The quickest way to select them is to use the Intersection option and select an enclosing rectangle.

Head merging

The JOINWALL command joins two lines of a wall. The command starts by asking to select the first line and then the second. The first line selected determines the starting point for the connection line, which is created perpendicular to the first line and ends at the intersection with the second line.

The program allows you to select walls not belonging to the current layer, though the walls selected must belong to the same floor.

Constructions with triangles



Trace just one side with the triangle method

The command TRIANG1 allows you to trace the side of a room starting from a side of reference selected in the drawing and from lengths of the side and of the diagonal which it forms with the existing one. The command first of all asks to indicate the layer of the new line which in general is placed on a layer of the walls.

Command: TRIANG1

Specify layer name for the newlines

<ADDC2dwalls\$0>:

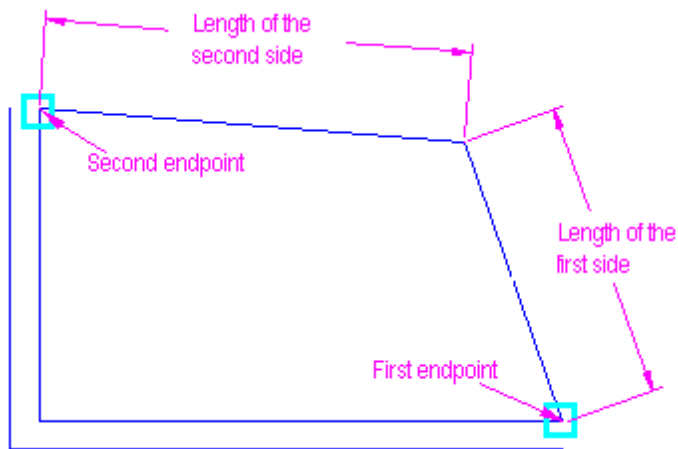
Select the line of the existing side:

Length of the new side:

Length of the diagonal:

Specify a point on the correct side:

As it is possible to draw either side of the existing line, you are asked to indicate a point to on the side you require.



Trace two sides with the triangle method

The command TRIANG2 allows you to trace two sides of a room starting from two points of reference, the distance of which generally represents a diagonal of a four-sided room and of the length of the two sides. As for TRIANG1, the command first requests the name of the layer upon which to create the entities.

Command: TRIANG2

Specify layer name for the newlines

<ADDC2dwalls\$0>:

Specify the first endpoint of the side that you know the length:

Specify the second endpoint of the side that you know the length:

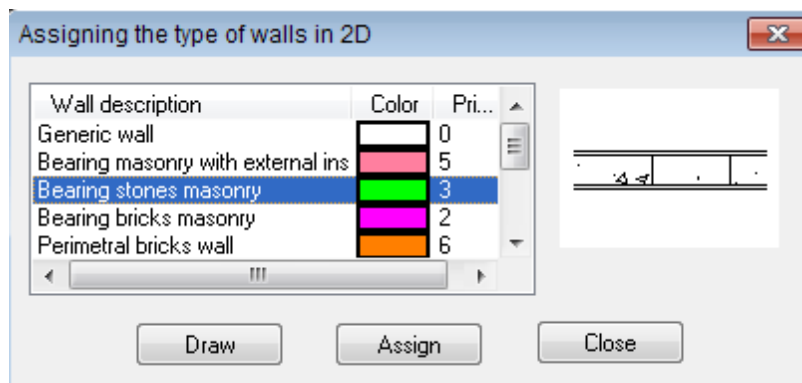
Specify the length of the first side to draw:

Specify the length of the second side to draw:

Specify a point on the correct side:

Drawing stratifications

The command DEFWALL is for drawing the layered representation of walls, obviously taking into account the position of the openings on the walls. It is also possible to define or change the definition of a wall, previously done with the WALL command.



Wall recognition

The command also recognizes walls which are generic and have variable thickness, as long as they are linear. In this version, the recognition algorithm only works with lines. The curved walls continue to have the two arc representation. Walls can only be recognized if they have a thickness less than the

value set in [Openings and walls 2D/3D tab](#).

Wall type list

The command has a dialog box with the wall type list, the image of the type of wall selected in the list and three buttons. The second column of the list indicates the color with which it is possible to recognize on the display which lines of walls are a certain type. The third column of the list indicates the priority associated to the type of wall. Both the recognition color and the priority of the type of wall can be modified by the [wall type definition system](#).

The *Draw* button is to draw the walls of the current layer with the representation assigned to each segment of the wall. Any pre-existing wall models are canceled before a new model is drawn.

Layering and current scale factor

The wall model obtained depends on the [current scale factor](#) set. For different scale factors there can be different hatches and a different number of layers. This depends on the definition of the wall type.

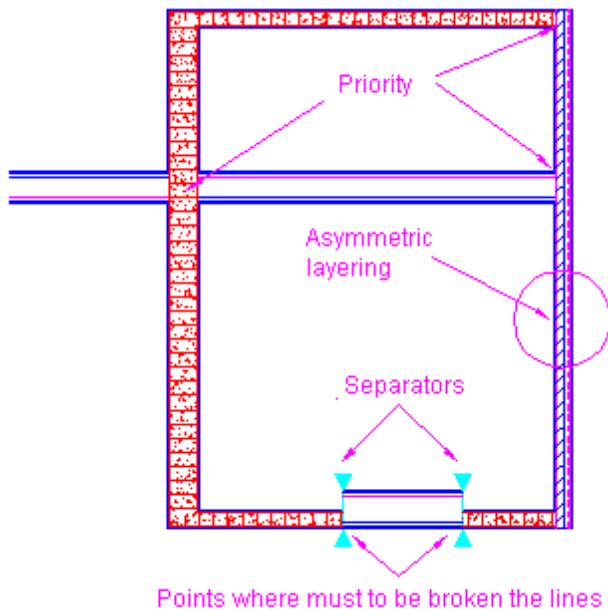
Layers used for stratification entities

Each stratification can belong to a different layer. The layers of the individual stratifications are decided when the wall is being defined. The entire stratification of a wall section becomes an anonymous block which can be exploded and manipulated like any block. The block is in turn placed on the ADDCSTRAT_BLOCK\$<FE> layer where <FE> stands for *Floor Elevation*.

Assigning wall type to drawing wall lines

The *Assign* button is for assigning the walls with the type selected in the list. Once you have finished assigning the type to the various walls of the floor and if you do not wish to draw the representation, you may exit the command by pressing *Close*. If you choose *Assign*, the dialog box is removed and you may assign the selected type to the various lines of the walls.

Stratification options



AddCAD allows you to deal with stratifications which are not symmetrical to the longitudinal axis of the wall. Often there are wall stratifications placed only on the outside of the building. Another problem which comes up when wishing to automatically draw the stratification of walls, is the rule which must be applied when two walls meet, for example in corners or intersections. Therefore you need to define the external side of the wall and a priority which governs what to do at intersections. To take care of this issue, AddCAD introduces the external attribute of a façade and the concept of global priority associated to the type of wall. Wall orientation can be changed with the command WALLATT which proposes the following options.

Command:WALLATT

Select the external side or [Remove external definitions/external Perimeter]:

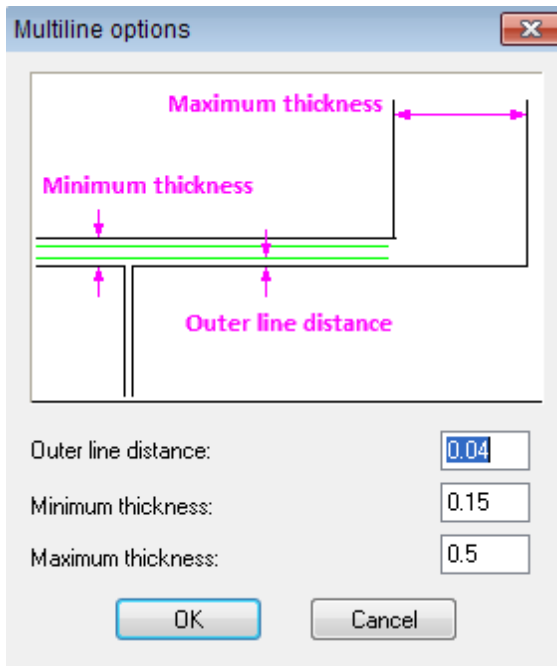
You can then select the external side of a wall. Eliminate the definitions of the external sides in the drawing and select a side which is part of the external perimeter. In this latter case, the external attribute is associated to all the lines of the perimeter.

When the command is launched, all the façades which have the external attribute are highlighted with a particular color in order to realize the current situation. The priority of a wall is a characteristic linked to its definition. The priority of a wall can be changed by using the [stratification definition system](#). Keep in mind that the higher the priority, the more that type of wall prevails in intersections and angles.

Separator object

The recognition of the wall at niches or in general where the thickness varies is only possible if appropriate separators are inserted. A separator object needs to be inserted as shown in the figure. Furthermore, the line must be broken at the points of intersection with the separators. The figure also highlights the role of the priority in the intersection and angle solution.

Multilines in walls



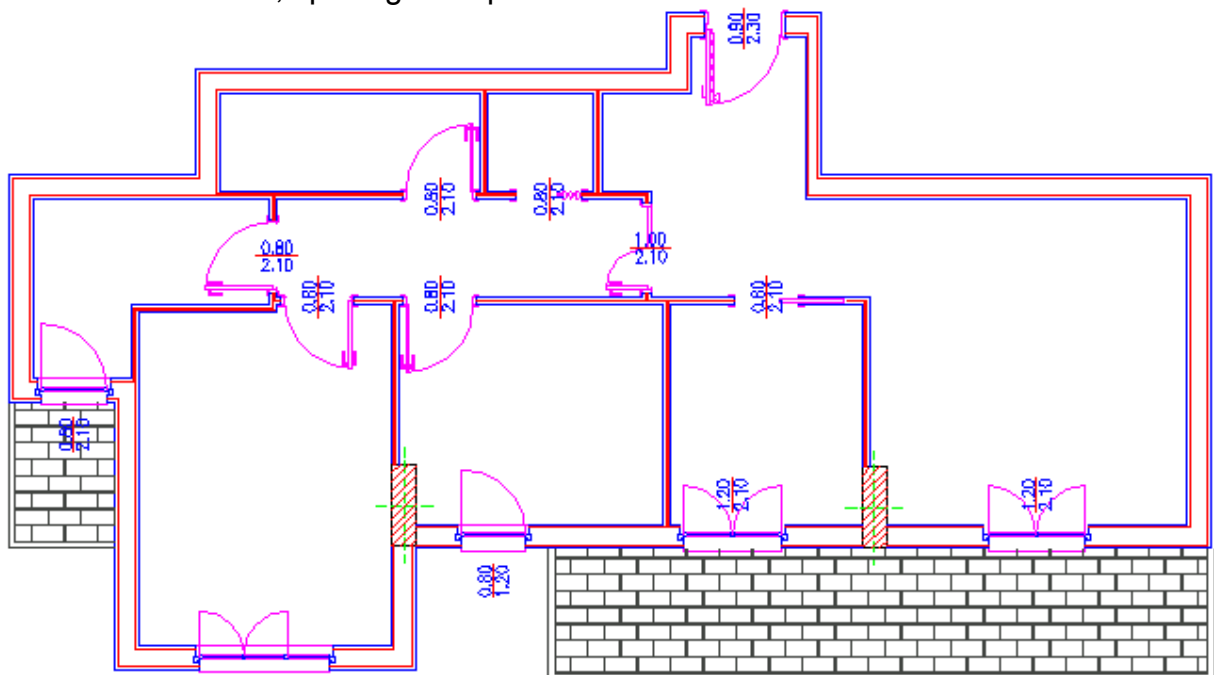
In some cases, the DBLWALL command can be an alternative to the stratification system. The command completely disregards the type of wall. It simply traces a pair of lines inside the wall automatically which can represent the coating of plaster or the wall cavity. The distance between the internal and external lines can be modified. The command displays a dialog box which illustrates the parameters used by the program.

It must be pointed out that only walls thicker or as thick as the minimum and less thick or as thick as the maximum are considered. If the minimum thickness is less than or equal to the smallest thickness of the wall and the maximum thickness is greater than or equal to the largest thickness of the wall, all the segments of the wall are processed by the command. The maximum thickness should never exceed the thickness of

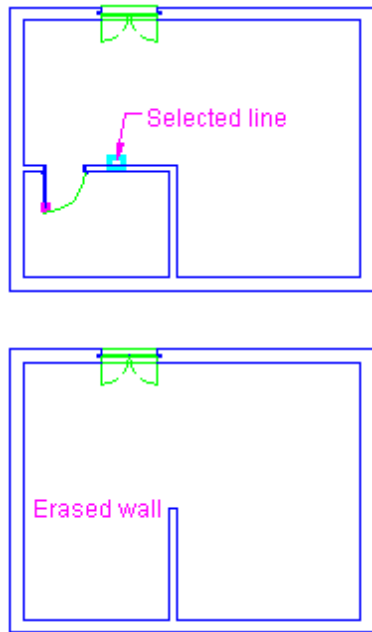
the largest wall of the drawing. The multilines are automatically broken at openings and pillars.

Automatic regeneration

The AddCAD model generators automatically regenerate existing models when started. This occurs with the 3D model, with the hatches, with wall stratification and with some calculus algorithms. The DBLWALL command is also capable of instantly updating the multiline model. It quickly corrects the drawing when modifications have made in the distribution of rooms, openings and pillars.



Erasing walls



The DELWALL command allows you to erase wall segments. All the openings and pillars which could be present in that wall segment are erased. The program attempts to fix all remaining walls.

The command requests the line segment to be erased.

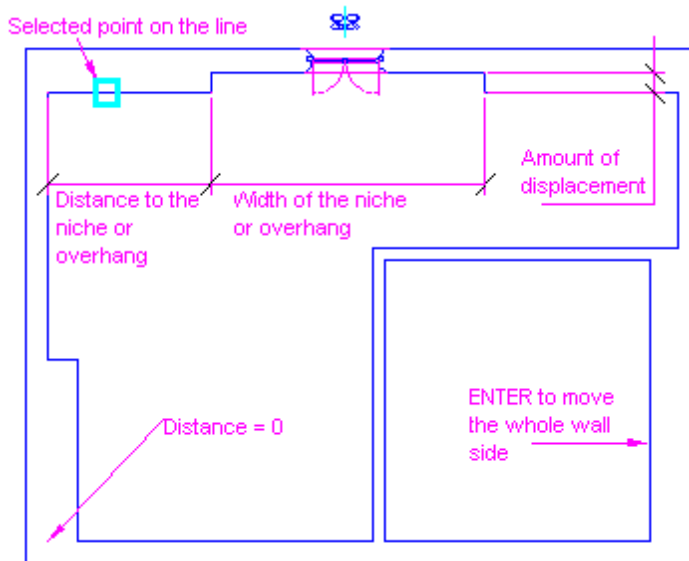
Command: DELWALL

Select the wall you want delete:

If the automatic 3D generation is active when a wall is erased, the modifications are automatically made on the wall 3D.

Modify walls

The EDITWALL command performs wall Editing, such as changing wall thickness, copying niches or overhangs due to the presence of chimney pipes and ventilation flues. The command requests you to select a line of a wall. *Pay close attention to the line selection point*, because the values of the distances requested afterwards refer to the endpoint of the line closest to the selection point.



Command: EDITWALL

Select a line of the wall side you want to modify:

*Distance to the niche or overhang/
Enter to move the whole wall side:*

If you answer this question by pressing *Enter*, the whole façade is displaced and therefore the thickness of the entire wall is modified. If you answer with a number or you indicate a point, then you are asked the width of the niche or overlap. Whatever you answer, the command will continue with the following request.

Displacement amount of the wall side or [Inside/wall Width]:

You must specify how much the overlap must come out or how much the wall thickness must be increased. You can alternate between displacement inside the wall (niche) and outside (overlap) with the *Inside* and *External* options.

Wall Width option

Allows you to directly assign the thickness of the wall and not to impose the amount of displacement of one façade with respect to the other.

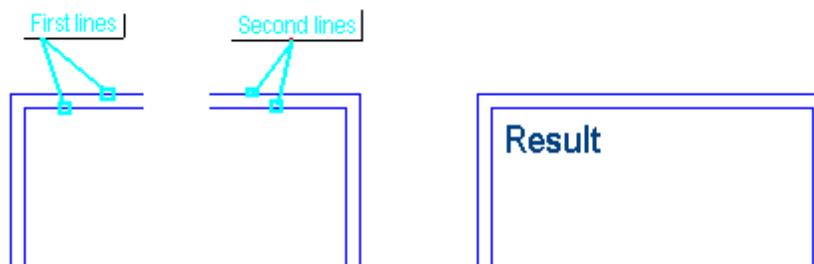
Wall width or [side Displacement]:

Limits of command

The openings found in the section of the modified wall are modified automatically. If an opening is only partially in the section to be modified, it does not lead to a correct result. Openings at the angle are not modified.

Join lines

This function is very useful when you wish to recover drawings made with ZWCAD alone. In these cases, the wall lines have been broken resulting from the insertion of blocks which represent holes such as doors and windows. In order to insert parametric objects, besides changing layer of the lines, you must join them where they were broken for holes.



Command: ULINES

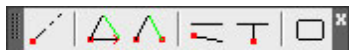
Select the first line to join or Enter to exit:

Select the second line to join with the first one:

Planimetric relief

AddCAD has a series of commands which allow you to draw a plan of buildings by means of a planimetric relief.

In the first stage rooms are drawn with the line command, the wall command and, much quicker when drawing four-sided rooms, with the RECTANGL command. Afterwards angular conditions and relief measurements are added on. The program does not actually compensate relief errors (almost no longer required due to the use of laser tools), but rather it modifies and corrects the layout traced in the first stage using standard triangulation methods.



Toolbar with planimetric relief commands.

NEWLEN command

The NEWLEN command allows you to assign a length to a wall façade.

Command: NEWLEN

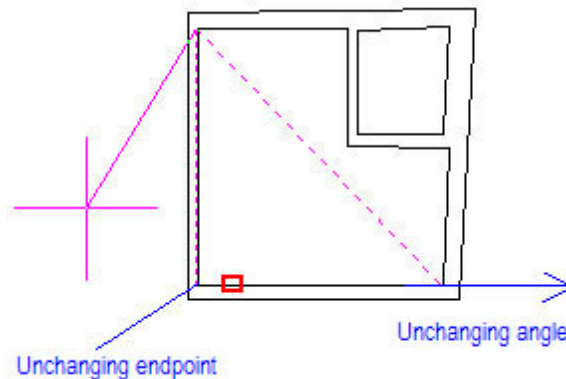
Select the line near the unchanging endpoint:

Newline length:<8.361>:

The command then displays the current length of the line in parentheses. You can enter a new value or press Enter to accept the old one.

You must pay attention to the selection point of the line. Select the line close to the endpoint which will not be displaced. The line angle does not change.

TRLINE command



The TRLINE command allows you to modify two lines by means of a triangulation.

Command: TRLINE

Select the line near the unchanging endpoint:

Select the second line:

Length of the highlighted line<4.899>:

Length of the highlighted line<8.017>:

Diagonal length<8.824>:

....

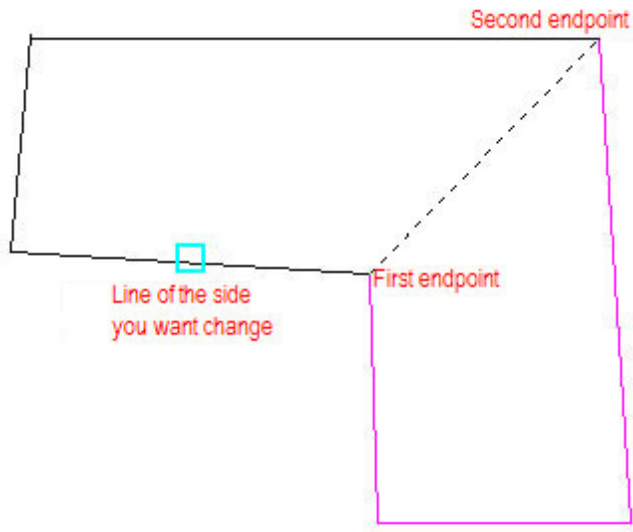
Notice the importance of the point selected on the first line. The point identifies the vertex of

the triangle which will not be displaced and unchanged. The first line selected also establishes that its angle will not be changed. It is not important where the second line is selected. The command then rectifies the lines based on the values entered.

Automatic stretching of vertices of other lines linked to the two triangle lines

A very convenient feature is that of automatic stretching of the endpoints of the other lines present and linked to the two lines of the triangulation.

TRPOINT command



The TRPOINT command allows you to modify a single line by means of a triangulation.

It is assumed to have a room similar to the one in the figure to the side and to have rectified the purple façades with the TRLINE command. And to wish to continue correcting the relief starting from the line indicated and from the endpoint vertices of the hatched line which we know are already correct. The TRPOINT command resolves this problem..

Command: TRPOINT

Specify the first endpoint of the unchanging segment:

Specify the second endpoint of the unchanging segment:

Length of the highlighted segment =8.824 (only for control)

Select the line of the side you want change:

Length of the highlighted line<8.017>:

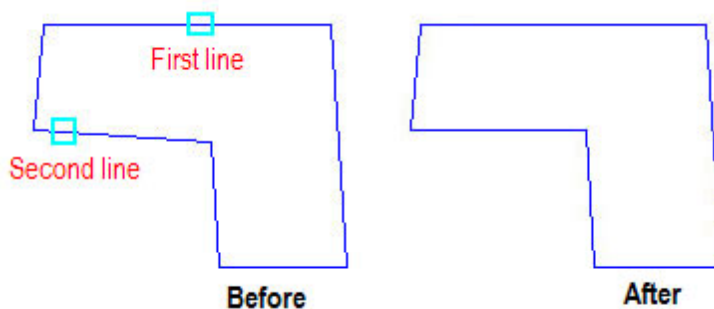
Diagonal length<4.899>:

The command then rectifies the line based on the values entered.

Automatic stretching of vertex of the other line linked to the line to be rectified

Another very convenient feature is that of automatic stretching of the endpoint of another line present and linked to the line of the triangulation.

PARALINE command



The PARALINE command allows you to modify a single line making it parallel with another given line. Looking at the example in the figure, you can see how the first line selected is the reference line. The selection point of the second line is important. The closest endpoint to the point of selection is taken as center of rotation and

therefore the position of this point does not change. You will notice that the length of the line to be corrected is not modified.

The requests of the command clarify what has been said.

Command: PARALINE

Select the unchanging reference line:

Select the parallel line near the unchanging basepoint:

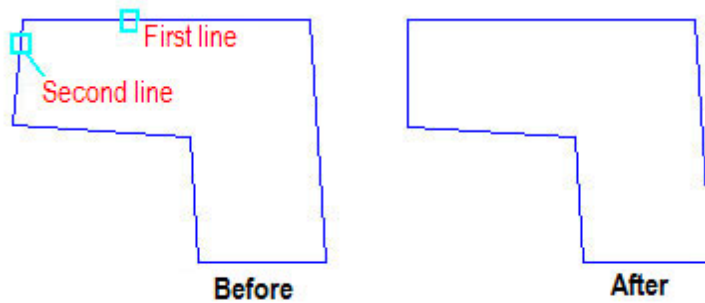
The command then rectifies the line making it perpendicular to the line of reference.

Automatic stretching of vertex of the other line linked to the line to be rectified

A very convenient automatism is that of automatic stretching of the endpoint of another line

present and linked to the line being rotated.

ORTHOLINE command



The ORTHOLINE command allows you to modify a single line making it perpendicular to another given line.

Looking at the example in the figure, you can see how the first line selected is the reference line. The selection point of the second line is important. The closest endpoint to the point of selection is taken as center of rotation and therefore the position of this point does not change. You will notice that the length of the line to be corrected is not modified. The requests of the command clarify what has been said.

Command: *ORTHOLINE*

Select the unchanging reference line:

Select the orthogonal line near the unchanging basepoint:

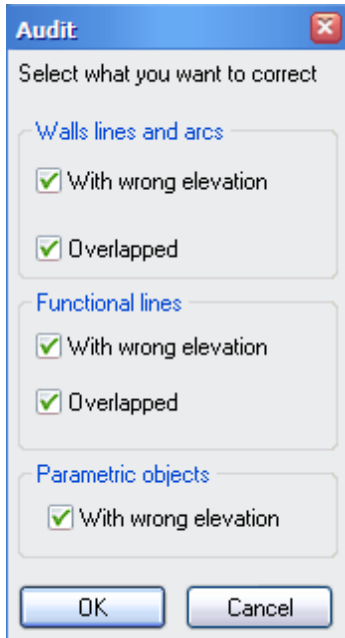
The command then rectifies the line making it perpendicular to the line of reference.

Automatic stretching of vertex of the other line linked to the line to be rectified

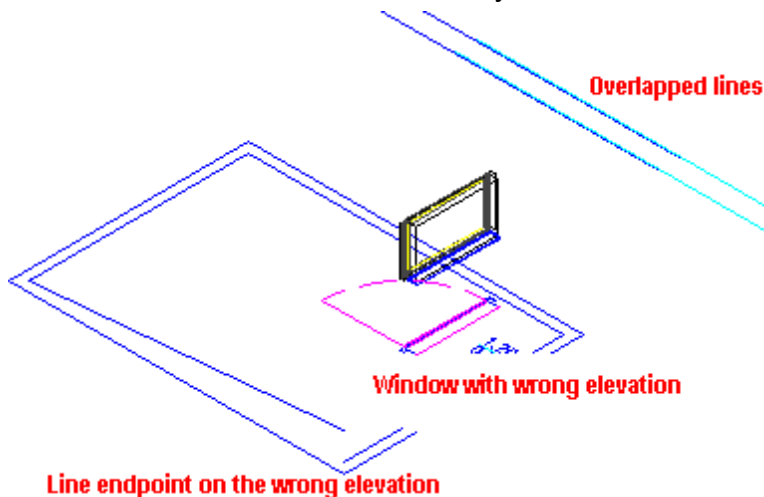
A very convenient automatism is that of automatic stretching of the endpoint of another line present and linked to the line being rotated.

Drawing audit

This command makes it possible to check whether the drawing has any errors and to correct them automatically. There are typically errors which prevent the program from working properly.



ADDAUDIT checks and if necessary corrects the following errors.



Walls lines and arcs, functional lines with wrong elevation

2D wall lines, arcs and functional lines whose parameters do not correspond to the layer they belong to (e.g.: line belonging to addc2dwalls\$0 with dimension z=2).

Overlapped Walls lines and arcs

Overlapping of 2d wall lines, arcs and functional lines belonging to the same floor.

Parametric objects with wrong elevation

Parametric objects with the block insertion point whose parameters do not correspond to that of the layer they belong to (same rule as 2d wall lines).

The command graphically highlights only the errors not corrected automatically, while it lists the number of errors found in the text window. We recommend using this command when unclosed contours occur when calculating room data.

Structural mesh

The MESH2D command allows you to insert a rectangular mesh containing the floor plan drawing. If you wish it can be measured with the [Vertices Dimension](#) command and obviously plotted. The dialog box of the command allows you to acquire all the information

necessary to create the mesh objects.

Mesh features

The elements of the mesh are placed on locked layers, meaning mesh objects cannot be moved, canceled or modified unless the relative layers are unlocked.

The layers used for the mesh have names characterized by suffix and prefix typical of [floor management](#):

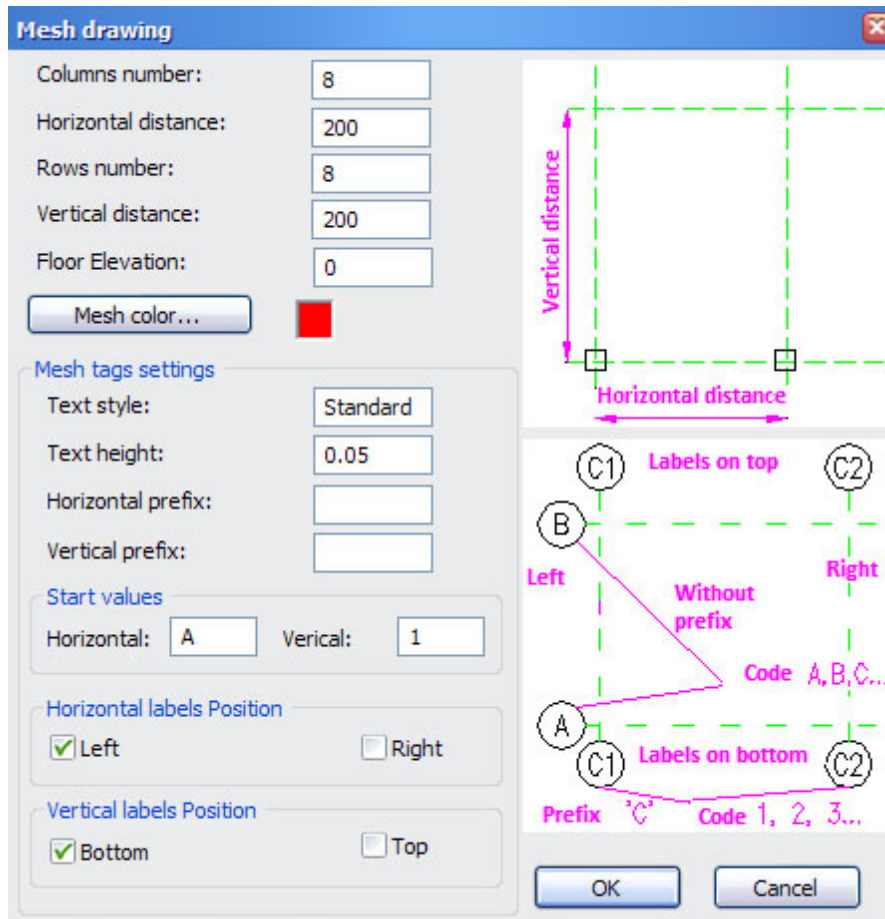
ADDC2DMESH\$<FE> for the lines

ADDCMESH\$<FE> for references to label blocks.

In turn the labels have 2DMESH_T layers for the texts and 2DMESH_C layers for the circles. The lines are the Layer type, and therefore it is possible to change color and line type with the [AddCAD layer configuration](#).

By the name given to the various layers, you can understand that it is possible to view the mesh only if the 2D plan view is active. Each mesh is generated on the floor Z elevation and can therefore be activated and deactivated in [floor visualization management](#).

Each line begins and ends in the intersection points. Therefore it is possible to indicate each intermediate point of the mesh with the Midpoint osnap.



Number of columns number of rows and distance

The number of columns and the number of rows of the mesh can be specified with different values for the vertical and horizontal distance.

Floor Elevation

The insertion on the current floor is proposed, but it is possible to specify a different floor elevation. In the latter case, if it does not already exist, the corresponding floor is generated automatically.

It is possible to specify the color of the mesh lines to distinguish it from others overlapping it which are on other floors.

Mesh tags settings

The style and height of the text of the tags can be specified. The tags can be placed as wished to the right and/or to the left and above and/or below the line endpoints.

It is possible to decide what type of progression to use to identify the horizontal lines and the vertical lines and also the point from where to start. If for example the letter D is placed

as the Start values Horizontal then D will be the lowest line, and moving up you will find E, F, G....

The prefixes of the tag text can also be useful, for example if there are several meshes in the same floor or if you wish to distinguish meshes on different floors.

Mesh insertion point

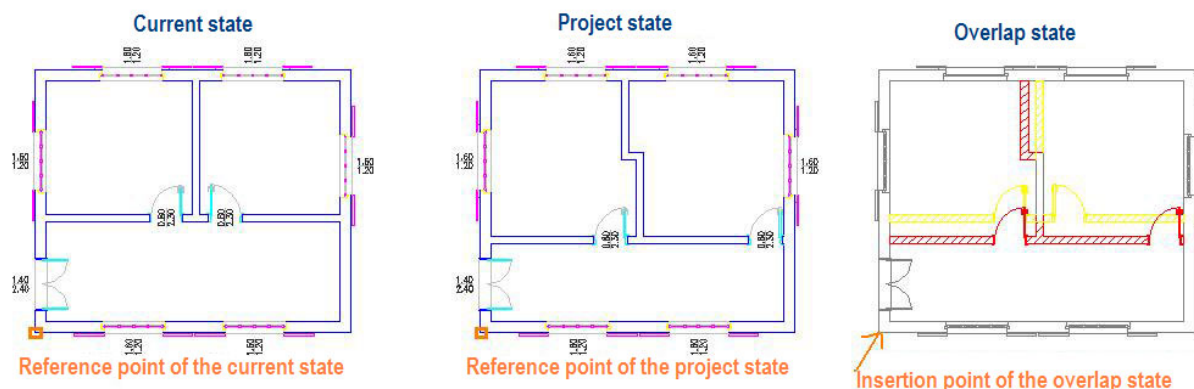
Before drawing the mesh, the program requests to indicate the point of the first node at the bottom left. You can respond by pressing *Enter* to have this node coincide with the origin of the axes (0,0). As previously said, the Z coordinate will always be the floor elevation specified in the dialog box.

Management of overlap states

In recent years, some of the most frequent activities in building are renovations and extensions.

In these cases, one generally starts doing a survey of the current state of the building. This survey almost always ends up making a plan drawing of the floors to be modified. These drawings identify and represent the so-called current state. Starting from the plans of the current state, normally from a copy, all modifications intended to be carried out are introduced. For example a window is eliminated, a wall is canceled, new walls and new openings are drawn until the drawing describes what we would like our building to become. These drawings are also called project states.

Further documentation comparing the two states is often required, showing details of all the proposed changes. This comparison comes about by placing the two states one on top of the other. This obtains the overlap state. In order to clearly express the extent of the intervention, the overlap state must be reproduced in three colors. Gray indicating whatever remains unchanged, yellow indicating elements to be demolished or openings to be eliminated and red indicating the new constructions or holes which need to be closed.



The OVERLSTATE command is capable of generating the overlap state based on the comparison between the current state and the project state.

The command requests first of all to select the current state, then the project state and finally the insertion point of the overlap state.

The reference points of the current state and project state are also requested. These two points identify the same position in the project. We can call it a point common to the two states.

The graphical result also depends on some option set with the [OSTATEOP](#) command. The requests of the command are therefore as follows.

Command: OVERLSTATE

*Select entities that represent the current state:
Specify the reference point of the current state:*

*Select entities that represent the project state:
Specify the reference point of the project state:
Specify the insertion point of the overlap state:*

Layers and overlap state

The program uses some layers where it places the entities of the overlap state, specifically:

2dosOpen_os, 2dosLine_os for gray openings and lines.

2dosOpen_cs, 2dosLine_cs, 2dosHatch_cs for yellow openings, lines and hatches.

2dosOpen_ps, 2dosLine_ps, 2dosHatch_ps for red openings, lines and hatches.

The doors and windows are still parametric. The wall lines go on the 2dosLine_xx layer.

Where xx stands for os, cs, ps, depending where they belong. In theory, placing these layers among layers for walls, the overlap state could be modified with the usual AddCAD commands. However it doesn't make much sense modifying anything in the comparison state.

Recognition of areas to be hatched

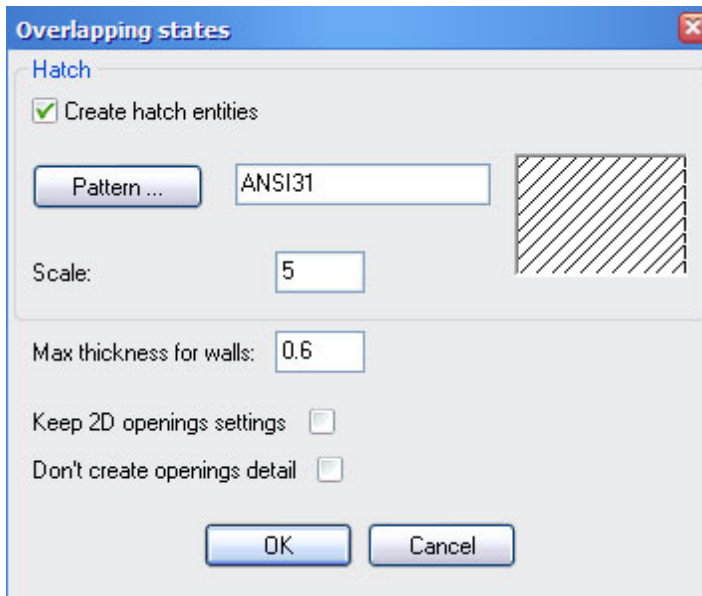
AddCAD always draws the openings and lines which correctly represent the walls in the overlap state. Often the areas to be demolished or constructed are also hatched. In some cases, the program is not capable of recognizing the areas to hatch. This is due to partial overlapping of walls and other causes. Since the lines are colored correctly, in some cases it is possible to intervene easily by hand with the AutoCAD and AddCAD hatch commands.

View of demolition and renovation plan

The OSTATEV command allows you to view at will only the yellow or red or both parts of the representation. The gray part is always left active. Thus it is possible to view the demolition and renovation plan separately. To separately print the two plans, we recommend > opening two viewports in layout space and to activate the relevant Overlap state in each one.

Overlap state generation options

The OSTATEOP command allows you to set some important options for [generating the overlap state](#).



Create hatch entities

If selected, the [Overlap state generation command](#) creates the yellow and red hatches relative to the areas to demolish and renovate. The hatch model and relative scale can be changed.

Max thickness for walls

The parameter is the same as in [Openings and walls 2D/3D tab](#) of AddCAD Options.

It is important in guiding the wall recognition algorithm. It should be increased if there are walls thicker than that indicated.

Keep 2D openings settings

If selected, the openings of the

overlap state are drawn following the [Openings drawing options](#) settings. If not selected, the openings are drawn without frame dimensions and codes.

Generation of the openings is always in the 2D representation only and never 3D.

Don't create openings detail

With this option it is possible to suppress generation of frame details (opening leaf, windows, etc) and to only view the holes in the wall of the openings.

Introduction

The parametric system is one of the most characterizing aspects of AddCAD. The parametric system makes it possible to insert objects which can be fully parameterized, change their dimensions at any time, manage types of objects and replace them. The parametric system is the cornerstone of the entire program.

Objects of the real world which AddCAD manages, such as openings, frames, balconies, stairs, gables, furniture, interior and exterior accessories etc. are parametric objects defined in just as many master files.

The insertion procedures and logical contents of the various objects can vary according to the category of the objects. For example, the insertion mechanisms of a window are different than those of gables. Even the associated logical information will be different for the windows which are light openings and the gables which must interface with roof slabs. Therefore there are different characteristics among the various categories of objects, but also similar procedures and mechanisms. The common features of the parametric objects make AddCAD much easier to operate and understand.

The AddCAD package includes a library of parametric objects sufficiently large for a most design in technical architecture. The library includes the following groups of architectonic elements: openings, balconies, roofs, external and internal accessories and furnishing, doors, windows, profiles, stairs, structures, pillars, sports facilities and worksite objects. The definition of these objects normally includes both a 2D plan representation, at times only symbolical, and a more complete 3D representation.

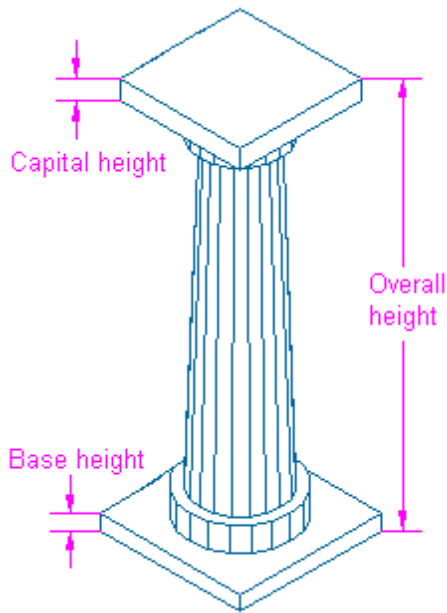
Measurement units used

Dimensional parameters must always be written in the unit being used. For example if you are working in centimeters, dimensions must be written in centimeters. It is possible to see and change the unit of measurement being used in the [Parametric and stair layer tab](#). It is not advised to change the unit of measurement more than once in the same drawing.

General considerations

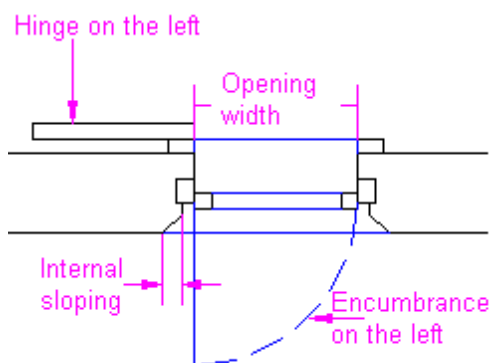
Drawing various graphical representations of objects can be speeded up and made easier by using a language which allows you to define object primitives and relative variable and constant parameters. This is the fundamental idea behind the definition of objects.

A complex object is made up of several entities (lines, circles, arcs, 3dfaces, etc). We can find similarities with AutoCAD blocks. Blocks are actually an ideal solution for some objects where there are three parameters (X, Y, Z scale) and where they are applied to the whole complex object. On the other hand there are cases, both in the architectonic field and in other application fields, where a drawing setting based only on blocks only resolves some problems but which more often has anomalies which greatly limit CAD potential.



Let us take a look at the example of a simple case which is quite typical in order to understand how the AddCAD parametric system works. Let's suppose that we want to draw some columns. Whenever you need to give it a different overall height, the problem cannot be resolved with a block, since for example assigning the Z scale with the value two, aside from doubling the overall height, it also doubles the height of the base and of the capital. This is not what you wanted at all, as you wanted them to remain as they were. The AddCAD object library presents even more important and complex examples. It's hard to think of using a block system to respond to the need of changing the side where the shutter and encumbrance of the window must be drawn.

Some applications have tried to provide solutions to these problems writing commands also called parametric procedures. Namely commands for building objects. In this case, for example, a command called column. But not even this attempt can lead to an acceptable solution as the types and number of parameters cannot be modified without rewriting the program code. Furthermore the case study of parametric objects is so large that it leads to extremely inefficient programs of enormous size.



Currently the most convincing solution is that of using a parametric language consisting in writing definition files of object prototypes, a solution also used by other CADs. The AddCAD parametric system, completely integrated with the application, besides being a simple parametric, also performs other functions (metric quantities survey system, automated opening insertion, information extended to opening vertices, fixed insertion points) unique to the application.

Parametric and Stair layer options

The Parametric and Stair layer Tab of the AddCAD Options dialog box allows you to

choose some options for the insertion of parametric objects. It is also possible to change the names of the layers of the stair modeler.

Drawing units

The conversion factor between the centimeter (unit of measurement used by the parametric system) and the unit used to draw.

Inter floor distance for parametric staircases

If deactivated, the stairs will no longer have this parameter but the step riser.

Text style for rooms and openings tabs

Here it is possible to write the names of the styles for the titles and data of the tabs and labels. The styles must be defined with the AutoCAD *Style* command.

2d representation for staircase

It is possible to choose the initial 2D representation to give to the insertion of the stairs. The representation can even be changed afterwards by selecting the single stairs.

Ask parameters

before inserting objects

Selecting this option allows you to avoid the dialog box to acquire the values of the parameters by going immediately to insert the object.

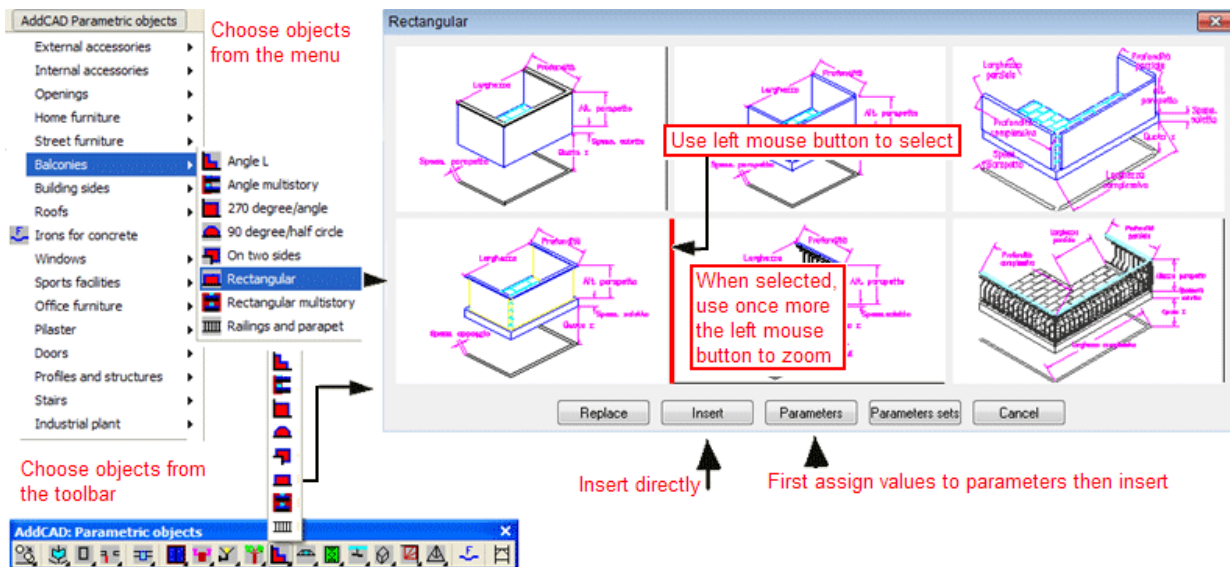
Size of fixed point symbol

The size of the fixed point generated with the [PFXEDON command](#).

Common features and commands

Choice of an object

There are various ways of choosing parametric objects. An object can be selected from the menus, an already existing object can be copied in the drawing or else you can use the OBJECT and OBJECTF commands. The insertion of a parametric object normally begins by selecting the object from the library menus.



You can choose the object both from the drop-down menu and from the toolbar. Once the selection window appears, you may select the object by clicking the image and selecting *Insert* or *Parameters*. Pressing *Replace* allows you to [replace the selected object](#) to other objects in the drawing and *Parameters sets* allows you to manage the types of objects.

Copy an object from the drawing

Another convenient method is that of copying the object by selecting one which is already in the drawing. This way it can be inserted immediately without passing through parameterization.

The other two methods of choosing an object require you to know the name of the definition file. The first is the OBJECT command and the other is OBJECTF.

Once the object has been chosen, you pass on assigning values of the object.

Command: OBJECT

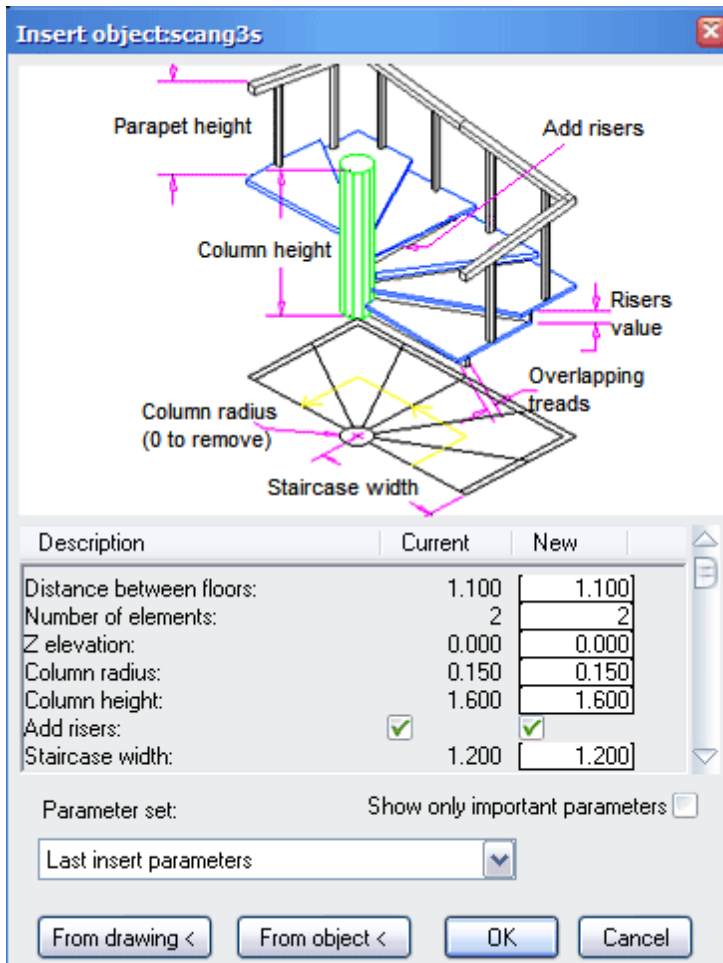
Object filename <last object>:

If you wish to insert an object which was already entered, just press Enter, otherwise write the name of the new object. As far as the name of the object file is concerned, a complete path must be written (without the .txt extension which is added by the program) or simply a name. In the latter case, the file must be present in the search path of support files. If the file is not found, an error message will appear. An error message can also be caused by the presence of errors in the definition file.

The OBJECTF command allows you to choose an object through the file browser of Windows.

Parameterizing objects

Once the object has been chosen, you pass on to assigning values to its parameters. The parameterization can be done inside the dialog box or on the command line. The second method is only used rarely and normally inside macros or Autolisp programs.



Parameterization through dialog box

Besides a help image to view the parameters of the object, there are the list of parameters, the list of types and a series of functional buttons. There is also a control box which allows you to customize details for parameterizing the object. Some parameters, once provided with standard values, do not need to be modified each time, whereas others, called *important*, are more susceptible to changes. Just activate this control box if you only want to view the main parameters to see immediately what needs to be modified. The parameters of the previous insertion are always proposed. It is not an actual parameter set. Once the parameter of an object has been assigned a value, it is statically memorized in the definition file and will always be re-proposed even in an following work session. This value, which we call **Last insert parameter**, will be modified if a

new value is assigned to the parameter. The parameter list has a three-column structure with an adjustable width. The first contains the description of the parameter, the second the current value of the parameter of the selected *Parameter set* and the new values you wish to attribute to the object in the third.

By selecting a *parameter set*, the list is updated, viewing the values of the parameters associated to the parameter set. If a parameter set is selected and the object is inserted without changing any parameter, the object will be created taking that *parameters set* into account. Then, for its future processing, AddCAD will take into account that it is an object with that *parameter set*. If one of the parameters of a parameter set is modified, then AddCAD will consider the object without a specific parameter set.

A parameter representing a linear dimension or a real value can also be modified by selecting two points in the drawing. The distance between them will take on the new parameter value. To temporarily hide the window and to acquire the parameter in the form of the distance between two points, just select *From drawing <*.

Some parameters, generally the Z elevation parameter, can also be acquired as coordinates of a point. When the value of the parameter is requested, you must respond on the command line with ".Z", ".X" or ".Y". You are then requested the point from which to filter the desired coordinate.

Pressing *From object <* allows you to assign the parameters to the new object by simply selecting one of the same type already present in the drawing.

Assigning parameters in the command line

This method is only used rarely and normally inside macros or Autolisp programs. The parameterization on the command line consists in a series of requests by the program with just as many answers to be written on the command line. It is possible to parameterize on the command line by setting the variable DLGOBJECT at 0 with the ADDCSETVAR DLGOBJECT command. Here are the requests.

Command: OBJECT

Object filename<last filename>:

Name of the parameters set or Enter to assign single parameters:

<Parameter description>: <last value>:

If you wish to insert an object with a certain *parameter set*, just answer with its name. Otherwise press *Enter* and then assign the values to the parameters one by one. The number and type of parameters varies from one object to the other. Some options are also available while inserting the parameters.

Copy the parameters from an object

Just press ">" when asked the value of a parameter. Obviously the selected object must be the same type as the one being inserted.

Object parameters request

Pay close attention to the message as there can be different types of parameters. For example we can have types of parameters for whole numbers, others for real numbers and still others for text strings.

The following are some typical request messages.

Sill elevation<0.900>:

Encumbrance on the right<1>:

Frame material<Wood>:

Once the parameter of an object has been assigned a value, it is statically memorized in the definition file and will always be re-proposed even in an following work session with AutoCAD. This value, which we call *Last insert parameter*, will be modified if a new value is assigned to the parameter. This value is also always proposed in parentheses at the end of the request message. The default value can be accepted by pressing *Enter*. If the parameters to be assigned to a new object are the same as those inserted previously, instead of pressing *Enter* for the amount of times equal to the number of parameters, just answer with "." when requested the value of a parameter.

Inserting objects

Once you have chosen the object with one of these three methods and it has been parameterized, you pass on to inserting it in the drawing.

The insertion of an object in the drawing varies depending on the category of the object. For example, the [insertion of a window or door](#) is normally done on a wall and therefore the position and angle of insertion are constrained by the position and shape of the wall. Gables are linked to the selected roof slab. On this page we see the insertion of unbound or generic objects. AddCAD provides you with a series of options to precisely and simply position the object in the drawing.

While the preview of the object moves on the screen as you move the mouse, the available options, using the right button of the mouse or the command line, are the

following.

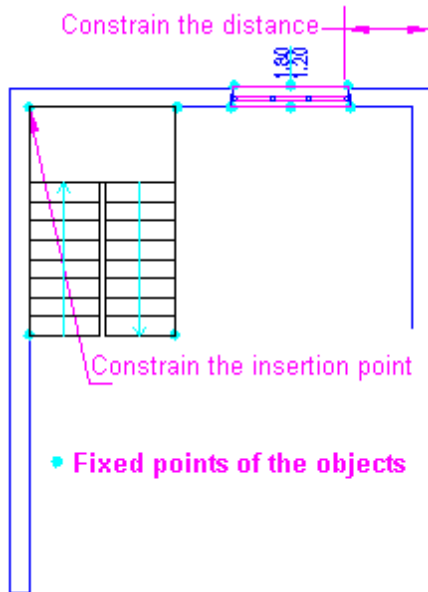
Insertion point or [Change fixed point/Modify parameters]:

Rotation angle <0.0>:

Insertion point or Enter to confirm the first one specified:

Modify parameters

The *Modify parameters* option allows you to go back to the parameterization box. If the moving image shows you that the dimensions of the object are not correct, this option allows you to assign correct values to the various parameters before actually inserting the object in the drawing.



Fixed points or important points of an object

A very useful function while placing an object is that of choosing the insertion point. With the option *Change fixed point* it is possible to choose which point of the group of fixed points of the object must match the point indicated for the insertion. The group of fixed points varies depending on the type of object and is set by the author of the parametric object. We recommend changing the fixed point with the F4 key on the keyboard or with the aid of the context menu. It is possible to know the fixed points of an object by repeatedly changing the fixed point until you return to the first one. When you change the fixed point, the cursor moves on to the image so that the intersection of the axes exactly matches the selected fixed point. The fixed point is also highlighted by a small red cross. One very important aspect of the fixed point, which also justifies its name, is that once the object is inserted, it

becomes a property of the object. Any modification made to the object keeps that point unchanged. If for example for a staircase we choose a fixed point at the top left and for a window the right shoulder, any modification of these two objects will not change the position of the chosen point.

Insertion with angle-point

The technique of inserting an object by first requesting the insertion point and then the angle of rotation is at times inconvenient and often forces you to make changes after the insertion. In fact the point-angle sequence is useful in just a few cases. Generally, since we have a moving image, it would be convenient to work with the opposite angle-point sequence. The AddCAD parametric system allows to decide which of the two techniques you wish to use, by just using the cursor for insertion.

After the request of the insertion angle, AddCAD requests the insertion point again. If you move the cursor, the rotated moving image moves along with the cursor. If you answer to this last request by pressing Enter, the object is inserted in the first insertion point and with the indicated angle. If you type another point, it will become the insertion point. This reverses the point-angle sequence.

Multiple insertion

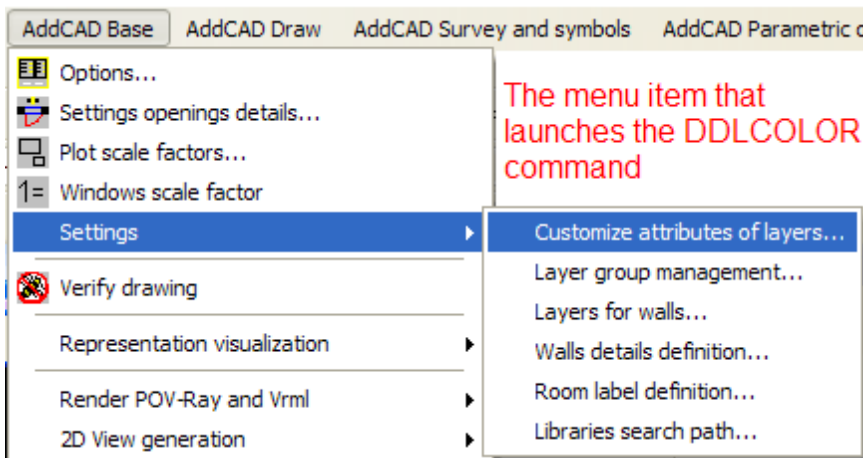
Some types of objects, such as doors and windows, can be inserted in a multiple ways. Once the parameters have been assigned and the first object inserted, AddCAD sets up for repetitious insertion, namely requesting to indicate the insertion point or to select a line of a wall once again. You can exit the cycle by pressing *Enter* with the keyboard or mouse.

Insertion dimension and Z Elevation parameter

Many objects, especially pillars, internal furnishing and staircases, are inserted in the [current floor elevation](#). The parameter *Z Elevation* which is often requested is therefore relative to the floor elevation. A value other than zero should therefore only be assigned if you wish to lower or raise the position with respect to the floor. Other objects not tied to floor such as parking lots, flower boxes, or outside lighting are inserted at z elevation=0, therefore the *Z Elevation* parameter means the absolute insertion dimension.

UCS and parametric system

The AddCAD parametric system, like many more of its functions, requires the global UCS.



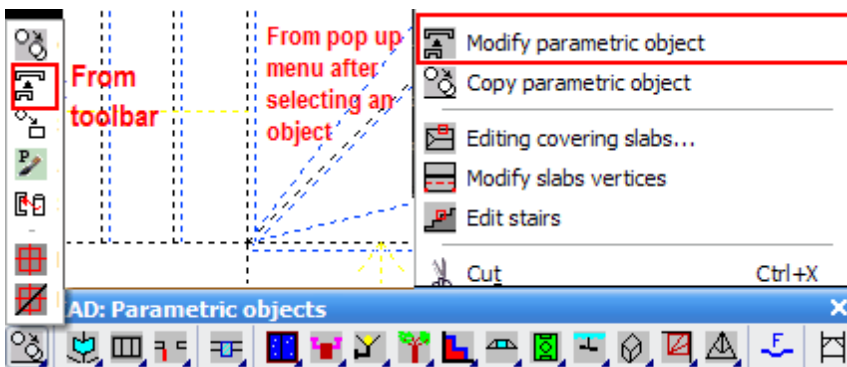
Automatic generation of layers and relative colors

The parametric system automatically creates all the layers specified in the object file, if not yet present in the drawing. The command generates layers with fully configurable color, lineweight and linetype. If you wish to automatically generate layers with different colors than those

intended in the standard configuration, you must use the [DDLCOLOR command](#). If you wish, it is also possible to have different names than those foreseen by the parametric objects for layers which AddCAD generates.

The automatic creation of new layers can be done in a way that the layers are ON or OFF. This depends on which representation is viewed when they are created.

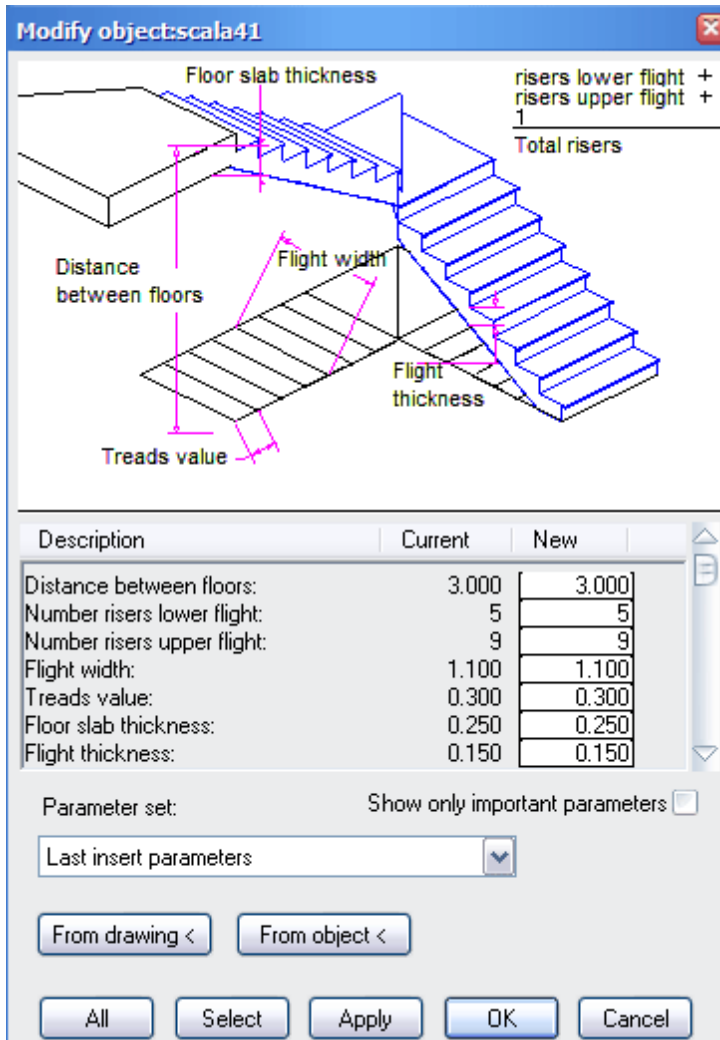
Modifying objects



The MODIFY command manages modification of the parameters of one or more parametric objects and the consequent regeneration. In the field of architectonic design, an activity often consisting in attempts, adjustments, variants etc., this command saves you a lot

of time. In fact the modification function is as simple as it is powerful. Just select the object

and immediately a dialog box appears which displays the current values of the parameters of the object. The structure of the dialog box is similar to the one for [inserting objects](#). The initial parameter list refers to the current parameters of the object. They can either be modified or a *Parameters set* selected. Once the parameters have been modified or a new *parameters set* assigned to the object, it is immediately regenerated. There are different modification options: you can modify only the selected object, modify all or only some objects. Once you have selected the object or one of the objects to be modified, the parameterization dialog box appears, also seen when inserting objects. The difference between the two dialog boxes is in the exit buttons at the bottom.



Apply

The *Apply* key allows you to modify the selected object. Without any further request, the object will be modified according to the new parameters.

OK

Often after having modified the parameters of an object, you also wish to change its position. This occurs whenever the transformations of the object cause it no longer to be inserted in the former insertion point. By means of this option, the parametric system allows you to reposition the modified object, choosing, if necessary, a different fixed point. Once the object has been regenerated, AddCAD requests the new point and angle of insertion, like for a new insertion. If you answer by pressing Enter, the position will not be changed. If you indicate the new position, the modification will obviously take the new position into account. In the case of modifications of openings on walls or other objects with constrained positions, the program

requests the new insertion point the same way as for insertion of a new object. This means that MODIFY allows you to close an opening in a wall and to open one on another wall. If you modify a gable, you are given the opportunity of moving it to another slab. If you modify a handrail, it can be associated to the opposite side of a flight of the same or another staircase.

Select

If you choose *Select*, you are asked to select all the objects to which you wish the modifications to be applied. The selected objects will be modified. In this case, it is not

possible to move the objects. The objects are modified each keeping its own fixed point still.

All

If you choose *All*, the program scans the entire drawing and modifies all the objects of the selected type by applying the following search criteria. Only objects of the same type with the same old values for those modified parameters are changed.

Automatic drawing updating

We've thought of associating a whole series of automatic drawing corrections to the MODIFY command. Currently the corrections made automatically are: adjusting lines and arcs of plan walls when an opening or pillar is modified, modification of hatches if present, modification of openings on slabs when modifying roof elements. Correction of the 3D wall model is performed by relaunching the WALL3D command.

Applying parameters

The PARAMAPPLY command allows you to quickly change the value of one or more parameters of a group of objects. The option *All* of the MODIFY parametric objects command allows you to modify all the objects of the same type (same object name) and that have the same values as the parameters to be changed. This command on the other hand, regardless of the names of the parametric objects, applies the value of the selected parameters of a source object to all the selected objects.

It is important that the objects have the identical parameter name, as "Door width" is different from "Opening width".

The command requests leave no doubts concerning function:

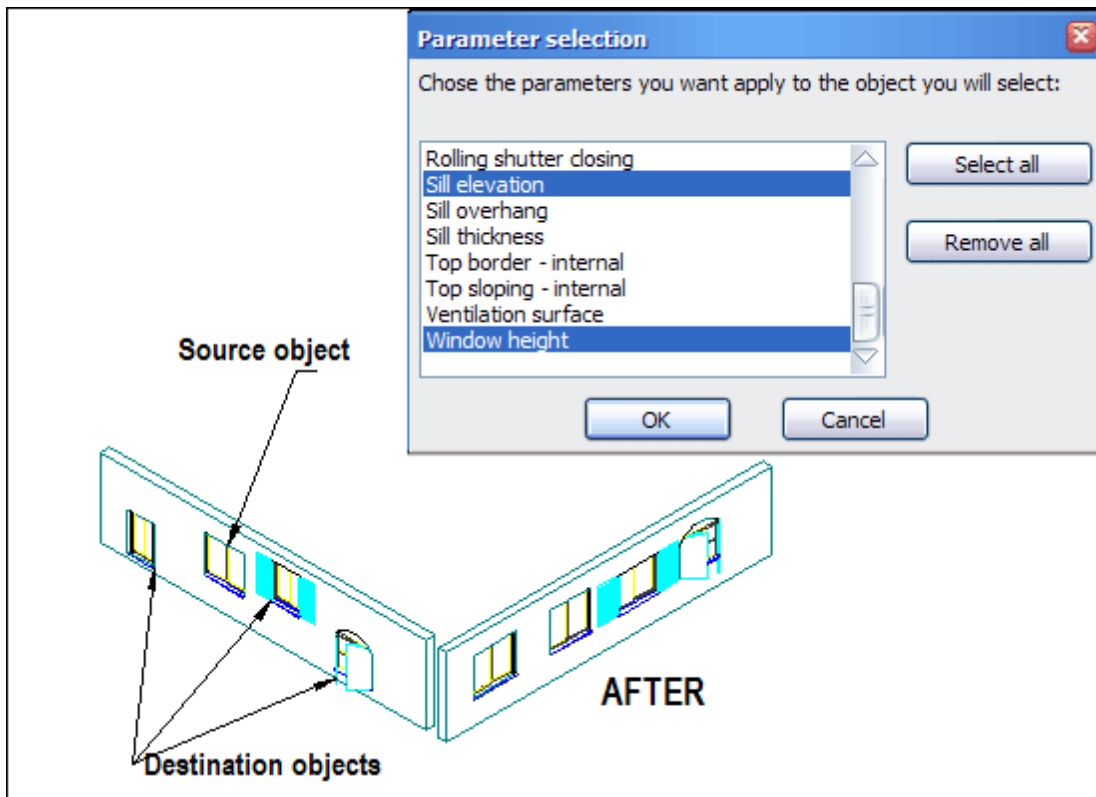
Command: PARAMAPPLY

Select the parametric object from which copy the parameter values:

<scelta dei parametri nella finestra di dialogo>

Select parametric objects to apply the new values:

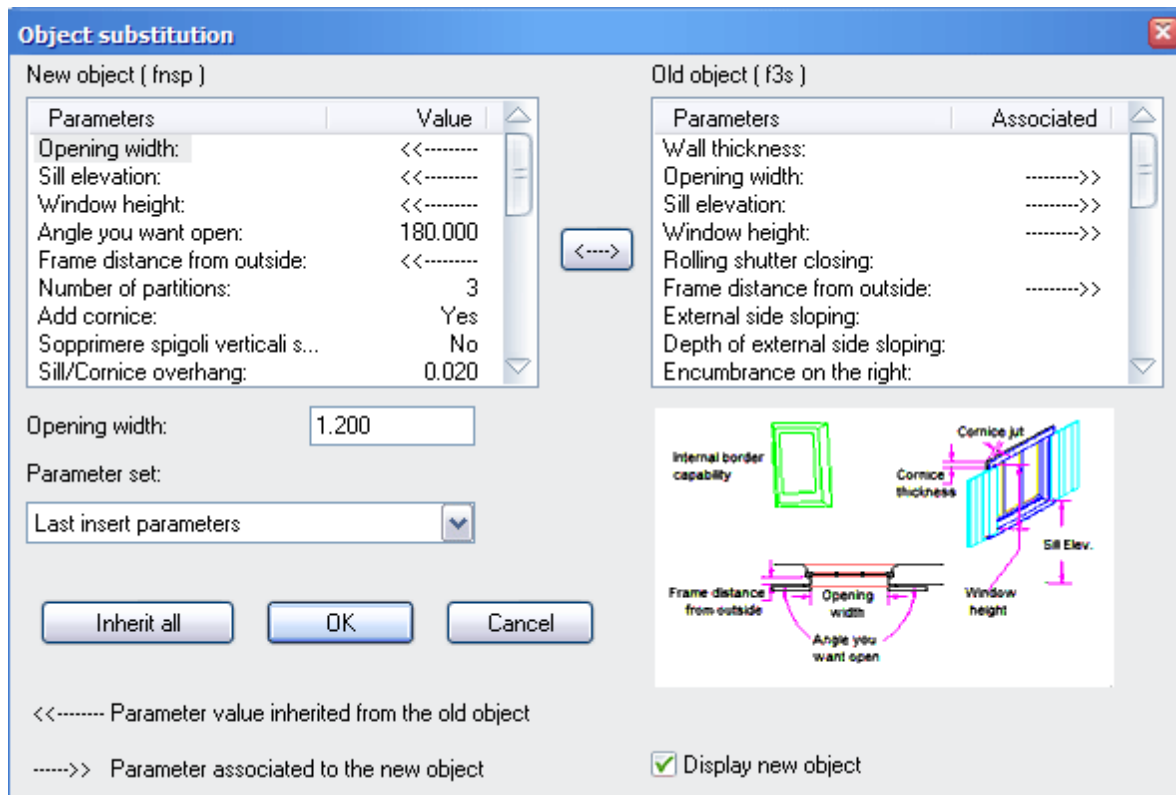
The following image clarifies the different execution phases. In the dialog box you can select all the parameters in the list or remove them with the relative buttons.



Replacing objects

If in the [window where the object to be inserted is chosen](#), we select an object and close the window by pressing *Replace*, AddCAD understands that we do not want to insert a new object, but rather to have it replace one or more objects in the drawing. In fact the next request regards selection of all the objects we wish replaced by the one selected. Once the selection has ended, AddCAD displays the substitution dialog box. This dialog box allows you to manage the association of the parameters between different objects and to assign new values by hand or by parameter set.

With the SUBSTF command, you can choose the new object to replace even by selecting the definition file directly from the folder of the hard disk.



This command is extremely useful when you wish to replace similar objects. For example replacing a staircase with another type of staircase, a window with another type of window or door, etc. In this type of replacement, the parameters are almost all automatically associated already and manual intervention is minimum. For substitutions between unlike objects, manually associating and applying values could be less convenient than canceling the old object and inserting a new one. In some cases, substitutions between different types of objects are denied by the system. It's hard to think of replacing a staircase with a shelf or a window with a chair. Therefore the command is not enabled for substitutions between unlike objects.

Old object and New object

The viewport at the top right has the description of the parameters relative to the object selected in the drawing, the viewport at the top left has the description of the parameters of the new object to which the parameter values of the selected objects are associated. Automatic association is done based on the comparison of the parameter descriptions. Furthermore, if in the example in the figure we have selected two or more openings, each with a different height, then each substitution will inherit its original value. If the right column has dashes instead of a parameter, it means that no automatic association has been made for that parameter. The new object probably does not have that parameter. In the list of the new object parameters (on the left) there can be both an explicit value assigned manually in the underlying edit box and the symbol <<-- which indicates that the parameter is associated and that the value is inherited from objects replaced bit by bit.

If you wish to associate parameters by hand, select one parameter from each of the two lists and press < --- > in between the lists. By selecting a relative *Parameter set*, you can also attribute a certain parameter set to the new objects.

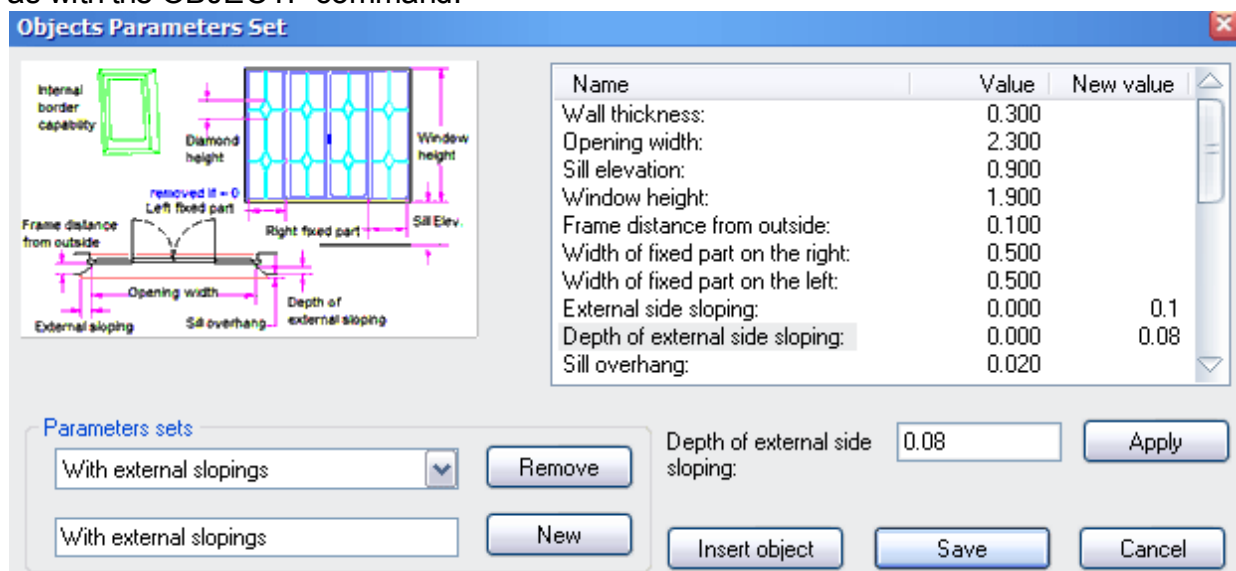
Inherit all

The *Inherit all* button allows you to reestablish the automatic default association.

The control box *Display new object* allows you to view both the image of the new and old object.

Parameters sets

It often happens that an object is used and parameterized in a certain way and then you wish to use it further on in the same project or in a future project. If in the meantime you decide to insert the same object with different parameters, the previous parameterization is lost. If you wish to use an object with two different parameterizations, different parameters must be assigned to the object each time. To tackle this problem, we have defined the *Parameters Set*. We consider a *parameter set* nothing other than a group of values to associate to parameters. To use a parameter set, it must be given a name. You will have noticed in the [dialog box for object selection](#) the presence of the *Parameters Set* key. If we select an object and then we exit pressing this key, AddCAD understands that we do not wish to insert a new object, but to add, modify or eliminate the *Parameters Set* of the selected object. Managing the *Parameters Set* of an object can also be started with the TIPOF command which selects the object through the file search, the same way as with the OBJECTF command.



The image of the object is reproduced at the top left. At the bottom left we find the list of *Parameters Sets* defined in the definition file of the object and a field to write a new parameter set name. A new parameter set can be added by writing its name at the bottom left and pressing *New*. Pressing *Remove* allows you to cancel a parameter set from the definition file of the object.

The list of parameters at the top right contains all the parameters of the *Parameters Set*. In order to modify the parameters of a *Parameters Set*, select the parameter set from the list even if it is new. The value attributed to a parameter of the *Parameters Set* can be modified by selecting it and updating the value in the edit box of the parameter. As the new value is being written, *New value* will appear in the column. Therefore you do not need to press *Enter* after having written the new value. The *Apply* key changes the definition of

the parameters set.

Save

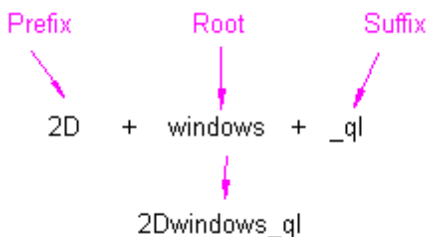
All modifications of the parameters sets can be memorized by pressing Save.

Insert object

Not only does it memorize the modifications of the parameters sets, but it also inserts the object.

Block and entity layers

A first distinction between layers, generated by the insertion of objects, is that of block layers and entity layers. Entity layers are layers where graphical entities are placed, for example lines, 3Dfaces, etc. An important characteristic of the object library is the automatic generation of numerous layers upon which various object entities are drawn. Then by being able to associate and modify both the color and material of the layer, you achieve the result of a rendering very close to reality and that of managing the widths of the pens associating them to colors.



At this point the layer must be given a name to make it easy to manage and recognize. The names of the entity layers of the objects respect the following rule. Layer names are composed of three parts.

Prefix

Indicates whether the entity belongs to a [2D plan or 3D representation](#). Two types of prefixes are used: '2D' and '3D'.

Root

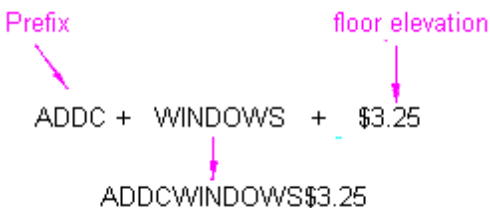
Indicates the type of object. Examples of roots are "stairs", "doors", "balconies", etc.

Suffix

indicates a component or a detail of the object. There is not always a suffix. When there is, it is separated from the root by an underscore '_'. It normally is composed of an abbreviation from one to three letters. Examples of suffixes are "_d", "_co".

This type of name allows you to easily use wildcard characters to set filters on layers, both to view the 2D and 3D model and with layer generation groups.

One way of knowing what a certain layer of an object contains is uploading the object onto an empty drawing and, among the generated layers, to only leave that which concerns us active. This allows us to see only what is present on that layer.



Block layers

Besides placing the single entities on different layers, AddCAD places the entire block on a layer called block layer. The name of the block layer is specified in the definition file of the object. For objects [linked to floors from a logical viewpoint](#) (such as openings, stairs, internal furnishing, etc..) the

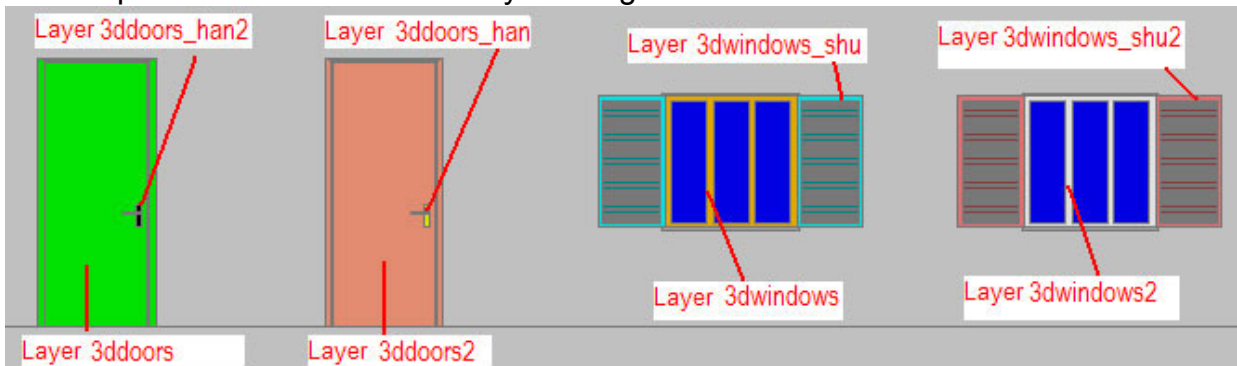
name of the layer is obtained by joining the symbol **ADDC** with the specific name of the definition file followed by the character "\$" and the value of the floor elevation of the insertion floor. For example, if in the definition file the specified name is DOORS and the floor elevation is 0, then the name of the block layer will be ADDCDOORS\$0. This

mechanism allows multi-floor designs to be managed very efficiently. For objects detached from floors from a logical viewpoint (such as gables, external furnishing etc.) the name of the layer is simply that specified in the definition file.

Changing layers to entities of an object

The viewing and rendering functions often require the use of different colors and materials for some object entities of the same type inserted in the drawing several times. [AddCAD generates an object](#), for example a door, always using the same layer called 3ddoors for the leaf. This prevents the viewing and rendering of the same type of door in the same drawing with different materials and colors. The same can occur with other 3d elements such as slabs, railings, etc.

The OBJECTCL command changes the name of the layer to the entities of a 3D block. For example referring to a door, all of the entities on the 3ddoors layer can be assigned a new layer called for example 3ddoors2. The next figure shows two examples of objects, a door and a window, which though being the same parametric objects, have a different visual representation thanks to the layer change.



The command asks to select the entity of the objects or of any block which makes it possible to identify the layer to be changed.

Command: OBJECTCL

Select an object on layer you want change the name:

The current layer is <3dgates_stru>. Specify a newlayer name:(esempio di nuovo layer)

Select all other object you want substitute the layer name:

Keep in mind that modification of the layer name on parametric objects is persistent, meaning that if I modify the object later on, the layers remain those assigned with this command. To return to the original layers, you must use this command once again and write the original name as the new layer. Other 3D blocks do not work the same way, as they are regenerated by setting the single commands.

Changing fixed points

The [fixed points](#) of parametric objects are important because they represent the points that do not change their position in the drawing after a modification of the object. An object fixed point can be selected when you insert or edit the object. In AddCAD 2012 version, we introduced the capability to [display the objects current fixed points](#). In this version you can also change the current fixed point at any time using the new command FIXEDPCH.

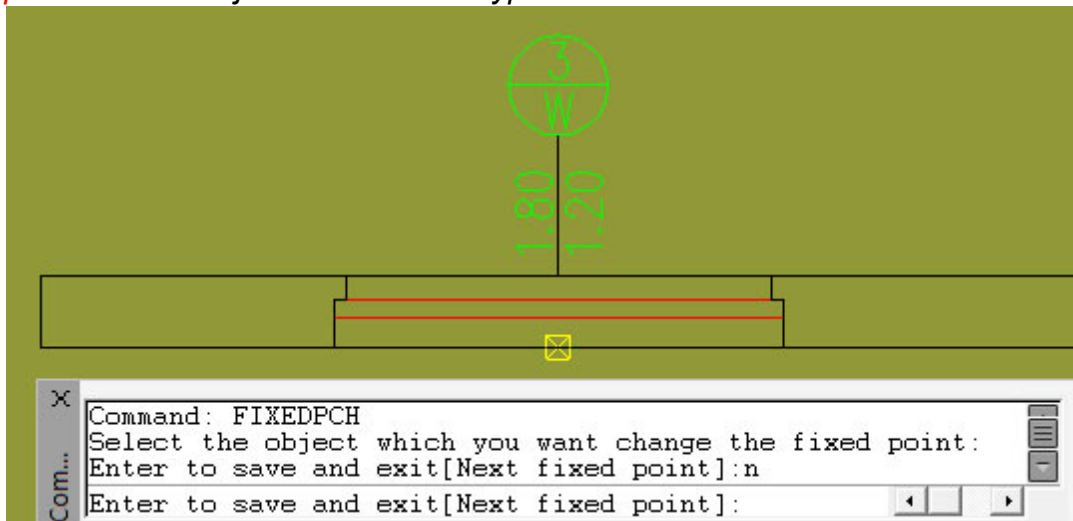
This command, once it has changed the fixed point to the selected object, allows you to select other similar objects to which you want to change the fixed point too.

Command: fixedpch

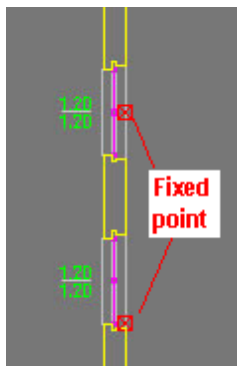
Select the object which you want change the fixed point :

Enter to save and exit[Next fixed point]:<Enter or N>

Select other objects of the same type, to which you want assign the same fixed point: <select objects of the same type>

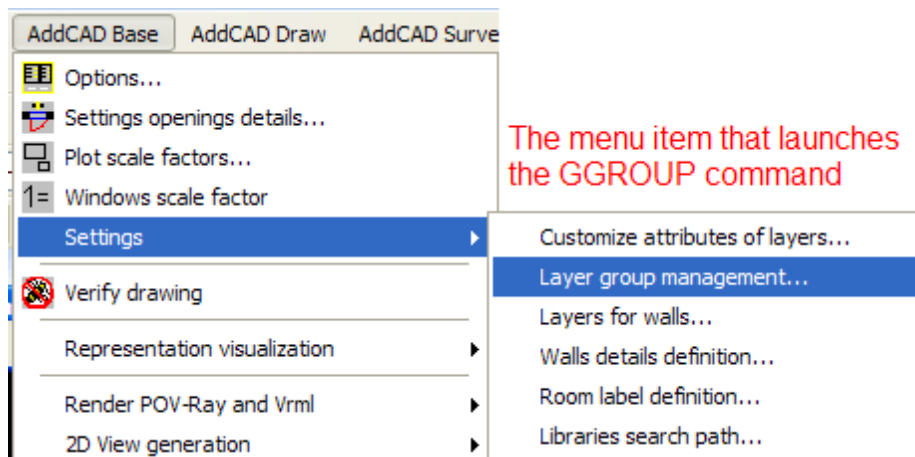


Viewing fixed points



While defining the project and especially when needing to modify or replace parametric objects, such as openings or stairs, it is important to know the "fixed point" of the object. This allows us to better foresee and understand the consequences of the modifications of the objects on the rest of the project. We have therefore defined two commands, FIXEDPON and FIXEDPOFF, used by simply clicking on the multifunction bar or toolbar. The user can configure a variable which adjusts the size of the fixed point. This variable is found in the [Parametric and stairs layer tab](#). This does not mean simply activating and deactivating layers. The commands intervene in the parametric object adding or eliminating the graphic indicating the fixed point.

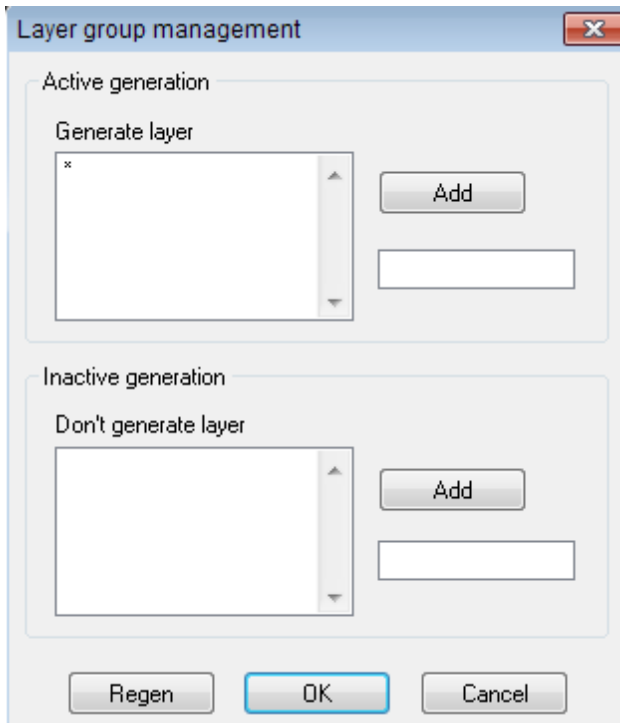
Different generations of objects



Each object can be generated in a drawing with a certain method. The AddCAD parametric system supports different generation methods by means of two mechanisms: [the scale factor](#) and the [layer mechanism](#). Each entity inside an object file is inserted

on a certain layer and each entity has a generation attribute which can depend on the plot scale factor in use. The choice of the representation to generate is not irreversible. The generation of objects in a drawing can be modified at any time. The current scale factor is chosen with the [SCALEF command](#) and the same command makes it possible to regenerate the representation at a different scale. Not all objects of the library support generation with different scale factors. These objects have the same representation, whatever plot scale factor has been chosen. To know which objects are capable of changing representation as the scale factor changes, see the description of the individual object groups.

The mechanism for generating different models based on layers is managed by the GGROUP command. This command allows you to configure layers you wish to create in a drawing. At any time you can generate all the entities, no entity or another group of entities. AddCAD gathers the entities in a multilayer object. All of the entities belonging to the plan representation are on layers starting with "2D", all of the entities with opening dimensions are layers which finish with "_Q", etc.



GGROUP allows you to establish which representation and which entities relative to layer groups are to be generated in a drawing or not. For example, we want to generate only the 2D model in a drawing, because perhaps we don't really care about the 3D or, on the contrary, in a different drawing, to generate only the 3D representation.

The layer inclusion and exclusion logic is resolved simply with two lists. It is possible to add a group of layers to both lists by writing it in the specific edit boxes *Active (Inactive) generation* and then pressing *Add*. And it is also possible to eliminate groups from the lists by selecting them one at a time and pressing *Del* on the keyboard. The descriptions of the layer can be used with special characters called wildcard characters. See the AutoCAD manual for a description of these

characters. The most important is definitely the asterisk, which substitutes one or more characters.

The generation logic is as follows: all of the layers present in the top list which are not in the bottom list are generated. These are the only rules for defining auto-generation. If for example I wish to generate only the plan representation, I can write 2D* in the list above or else leave the asterisk above and write 3D* below. Wishing to avoid generating, for example, the entities of the opening dimensions, just specify the *_d*. filter in *Don't generate layer*. And to avoid generating all that concerns stairs, just write *stairs* in *Don't generate layer*.

OK

The new button will be available on subsequent object insertions and on subsequent operations on objects. Objects already inserted in the drawing will not be modified.

Regen

The new setting not only has effect on new objects which will be uploaded but the entire drawing will be regenerated by updating all the objects. Take note that "*" is placed in the list of layers not to be generated. There will be no more entity of the objects in the drawing. At any time, the groups can be modified and the objects regenerated. This is due to the fact that AddCAD always keeps the information on the objects in the drawing.

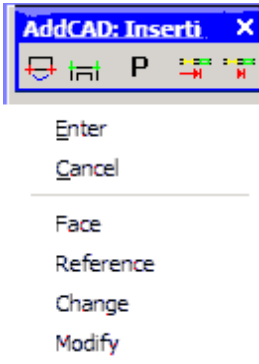
Openings, doors and windows

Insertion of openings

The insertion of these types of objects have more options and features than the [insertion of generic objects](#). Openings are separated in flat openings on a normal wall, flat and curved openings on the curved wall and at an angle. Although there are some small

differences, the insertion of these objects is very similar.

• Insertion of openings on linear wall



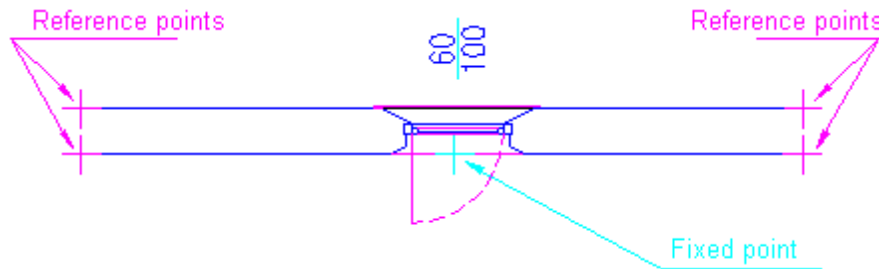
To make insertion of these objects easier, after the parameters have been assigned, the wall where to insert the opening is requested. In fact, for openings in linear walls, the message displayed is as follows.

Select the inside line of the wall or Enter to exit or [Modify parameters/Replace]:

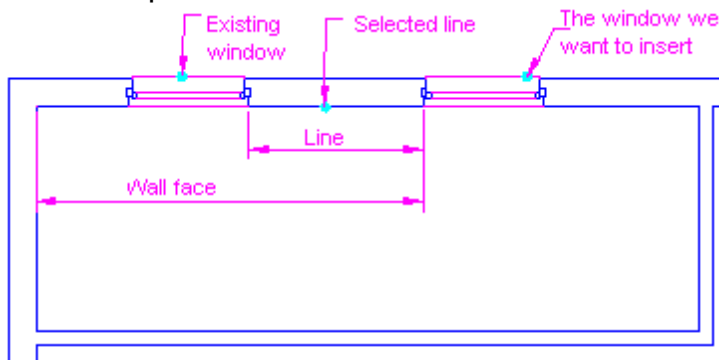
To select a line, any 3D view must be deactivated and the 2D representation activated. The entity selected determines the floor elevation and therefore the insertion level. If windows are inserted, the message is clear; if instead doors are inserted, you must select the line of the room on which the door is opened. To decide whether to choose those with the right side or left side hinge, you must keep in mind that the door is seen from the inside of the room in which it is opened. AddCAD automatically calculates wall thickness and once the object is generated, it constrains the opening to slide along the wall and requests its position in the form of the distance between a reference point and the fixed point of the object.

Distance or [wall Face/Reference/Change fixed point/Modify parameters]:

As usual the options can be recalled by right-clicking the mouse or on the following toolbar. The *Change fixed point* and *Modify parameters* options are the same as for generic object insertion. The first option allows you to change the opening insertion point, while the second opens the parameterization dialog box to change the value of the parameters.



Reference points



Four cyclically changing *Reference points* are available. The fixed points can vary from one object to the next, but they are generally the right and left shoulders and the midline both for the outside and inside.

The *wall Face/Line* option is for when there are several openings in the same wall. It

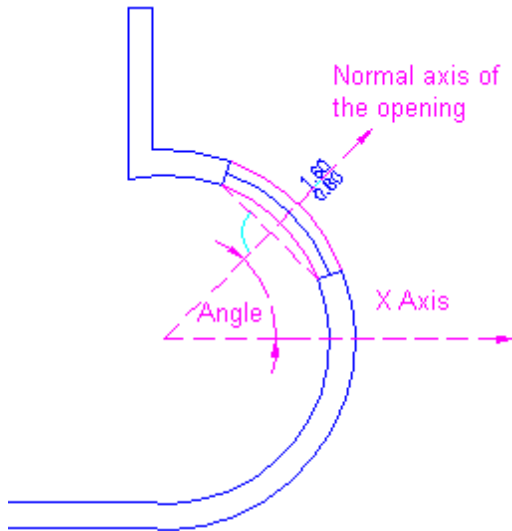
allows you to alternately move the reference from the endpoint of the wall face or to the endpoint of the selected line.

- **Openings on corners**

As far as openings on corners are concerned, the insertion point is very easy to find as the program resolves it, making the following request.

Select one of the two lines that forms the corner or [Modify parameters/Replace]:

Just select one of the two internal lines forming the corner.



- **Openings on the curved wall**

Openings on the curved walls, namely walls made up of double concentric arcs, are inserted by answering the following specific requests.

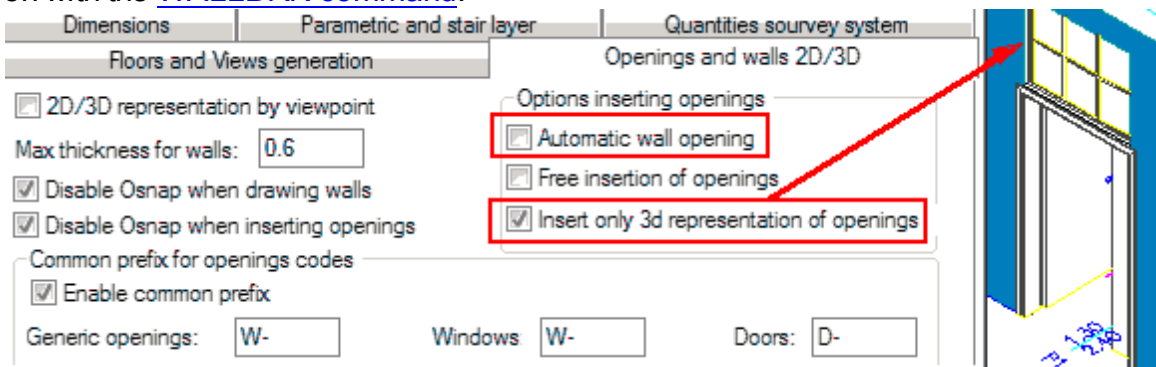
Select the inside arc of the wall or [Modify parameters/Replace]:

Insertion angle for the opening or [Modify parameters]:

Insertion angle means the angle between the X-axis and the normal axis of the opening.

- **Breaking wall lines**

When openings, pillars and frames are inserted or modified, normally the lines and arcs of the walls are automatically broken. In *Openings and walls 2D/3D Tab of AddCAD Options* it is possible to deactivate the Automatic breaking control button to keep AddCAD from breaking these entities. In this case, the lines and arcs can be broken later on with the [WALLBRK command](#).



- **Insertion of overlap openings**

In *Openings and walls 2D/3D Tab of AddCAD Options* it is possible to deactivate the 2D

generation and to disable wall breaking. This allows overlapping openings to be inserted easily on the same floor.

- **Free insertion of openings**

Flat openings have the possibility, by activating the specific option, of manually inserting the opening and therefore without selecting a wall. This will be useful when you need to insert a door between rooms only separated by furniture. In this case the request message is the following.

Select inner wall side or Enter for insert without wall:

If you press *Enter*, you will pass to free insertion of the opening by simply indicating the insertion point in the drawing. Remember that in this case the parameter 'Wall thickness' will also appear in the parameter list. This makes it possible to appropriately dimension the sides of the opening.

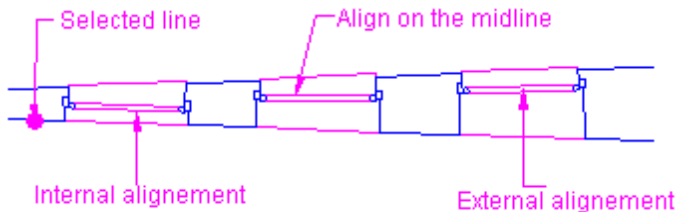
- **Common prefix for frame codes**

The openings normally have a parameter for the Frame code prefix. This parameter is linked to the sequential number of the frame thus creating the frame code. This further parameter has proved to be useless in most cases, simply adding to the list of parameters. If you decide to enable use of a common prefix for doors, windows and simple openings, you can establish a one-off common prefix for each of the redundant categories.

- **Automatic modification of the geometry of the frame in the variably thick wall**

AddCAD automatically adapts openings, doors and windows on walls with a variable thickness. AddCAD notices when a frame is inserted in a wall of variable thickness. In fact in this case an additional request is made.

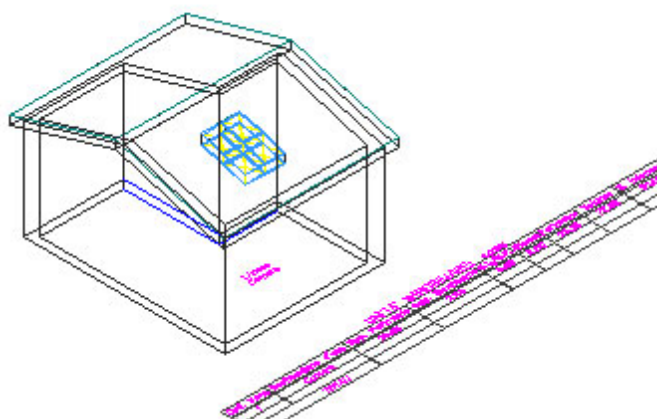
Select the alignment side or Enter to align on the midline:



You are requested to align the frame: externally, internally or to position the frame with the middle angle. You must select the line of the wall to which you would like to align the frame. A negative answer means that alignment will be performed in the middle position.

Skylights

Light elements on roofs.



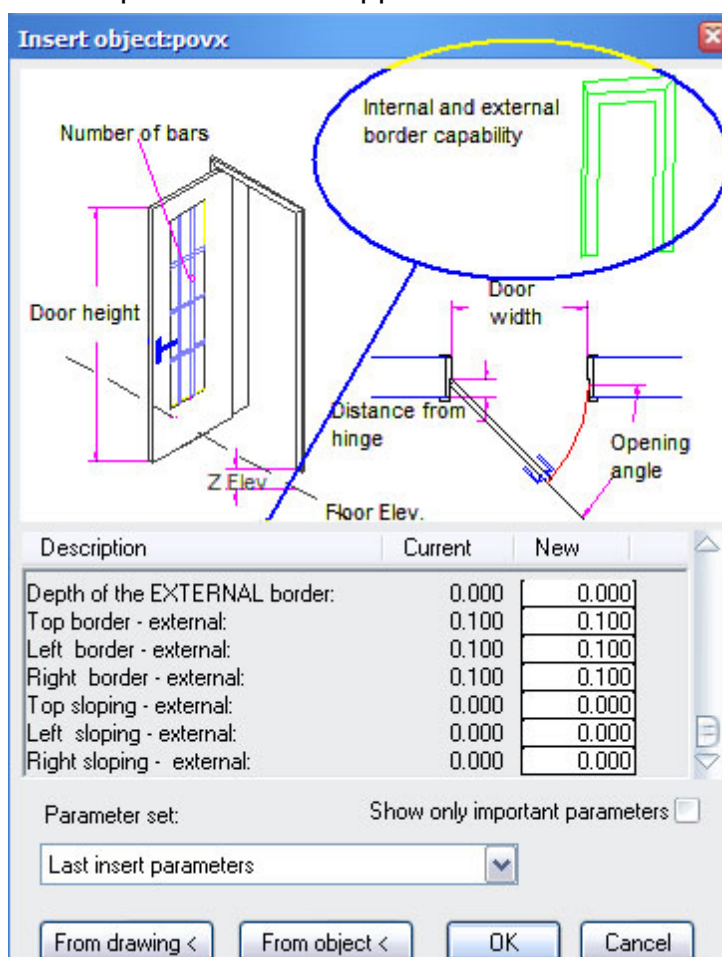
The skylights of the AddCAD parametric system add to the calculation of the [lighting area of a room](#).

These objects are linked to the floor. Therefore when they are inserted you must pay attention to the [current floor of the drawing](#).

Belonging to a room is detected based on the position of the reference point. Therefore if the reference point is inside the room, the lighting area is attributed to that room.

Wall border, sloping parameters

Many parametric windows and doors have parameters capable of implementing medium-large thick borders inside the walls. The parameterization window of a library door shows how the parameters are applied.



Symbol of openings with border parameters

Doors and windows with parameters which allow borders to be implemented in walls have a symbol in the image slide of the parameters. Windows normally have the possibility of implementing borders only inside the room, whereas doors both inside and out. There are two reasons why the slide does not indicate the explanatory dimensions of the parameters for implementing the borders. The first is because there are many of them and they do not fit in a small slide. The second is that they are the same for all objects.

Annulment function of "Depth of the border" parameter

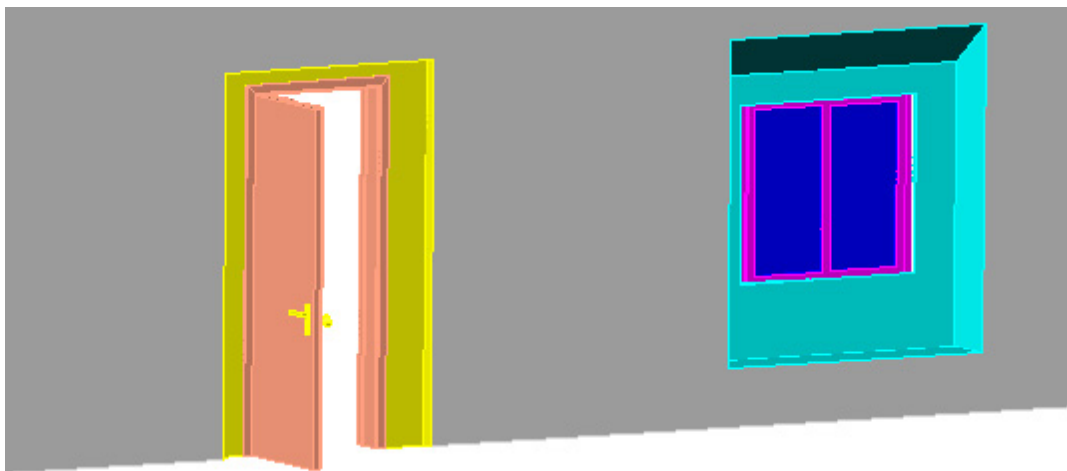
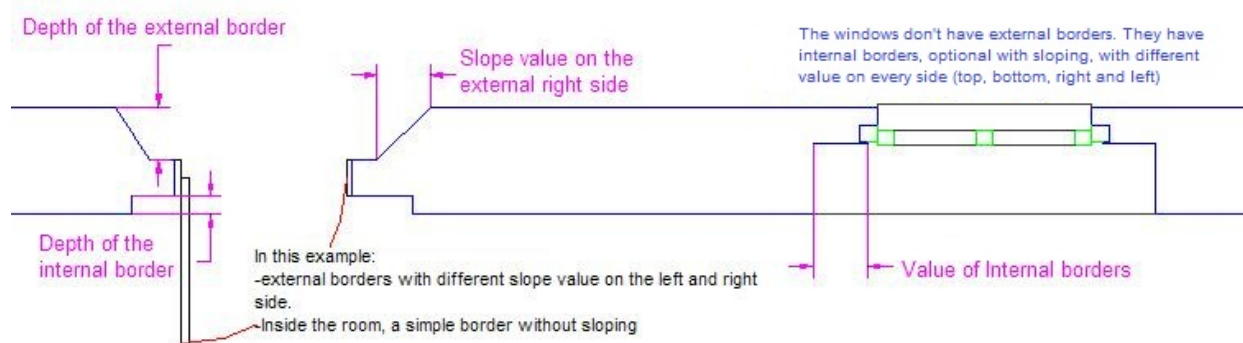
To simplify parameterization when there are no borders or slopes, the parameters *Depth of the EXTERNAL border* and *INTERNAL* are used to annul all the other parameters. For example, assigning the value 0 to *Depth of the INTERNAL border* means

assigning 0 to all the other border and sloping parameters. This way, by disregarding the value inserted in the other parameters, it is possible to quickly generate an opening without borders or slopes.

Layers used for border and slope entities

The line entities representing borders and slopes are placed in the *2ddoors_w* and *2dwindows_w* layers depending on whether it has to do with doors or windows. The 3D surfaces are placed on layers with the suffix *_ew* for external border entities and *_iw* for internal border entities. For example, the external border surfaces of the door are placed on the *3ddoors_ew* layer.

The following images illustrate the results which can be achieved with the parameters for borders in walls.



Wall break

WALLBRK is a command which is very useful in 2D wall drawings for breaking lines and arcs at doors, windows, pillars and openings in general. When openings and pillars are inserted, the lines are broken automatically by activating the control button [Automatic wall opening of Openings](#) and walls 2D/3D tab of AddCAD options. If you wish to break the wall later on, this command allows you to do it automatically. The command has two options.

Command: WALLBRK

Select an opening or an pilaster [Whole current floor]:

Whole current floor

If you type *Whole* the program will make no more requests. It will automatically break all the lines and arcs of the walls where openings and pillars are located. Only the entities of the current floor are taken into consideration.

Object selection

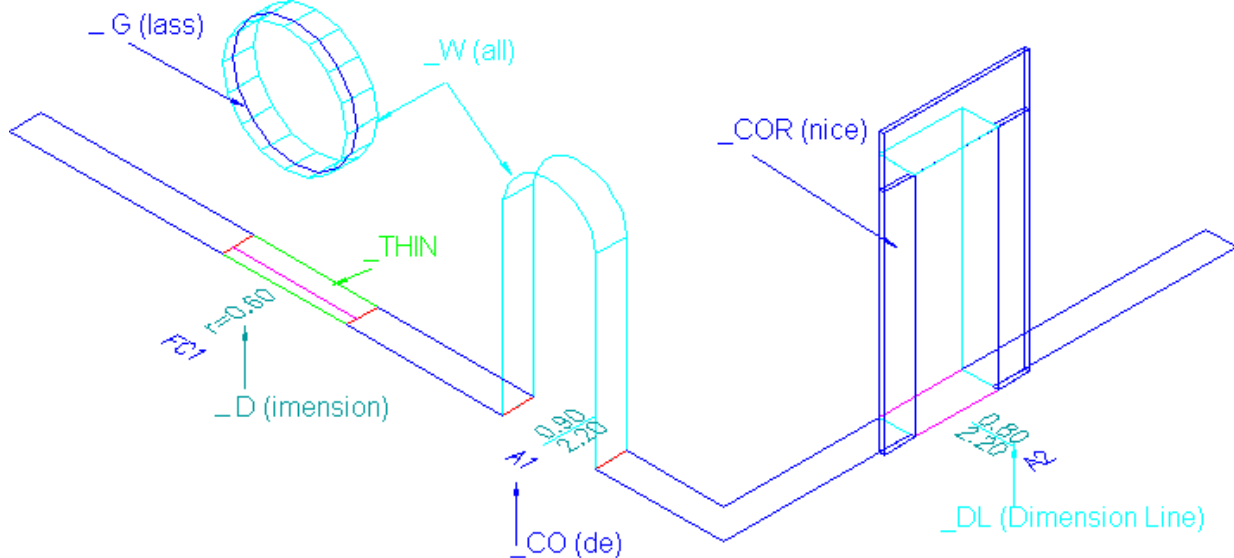
Only the lines upon which the selected object is located are broken.

The selected object must be a parametric opening or pillar and it does not matter which floor it's on.

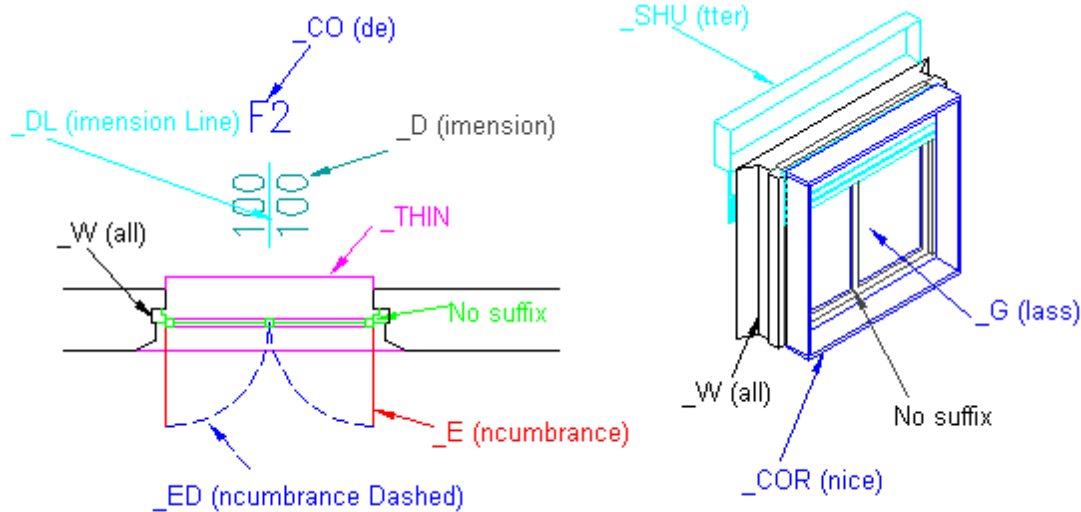
Layers of doors and windows

The following figures illustrate the suffixes of the most important layers for this type of objects. To understand more we recommend inserting an object and then activating or deactivating the layers or assigning them with different colors to see which entities are on the various layers.

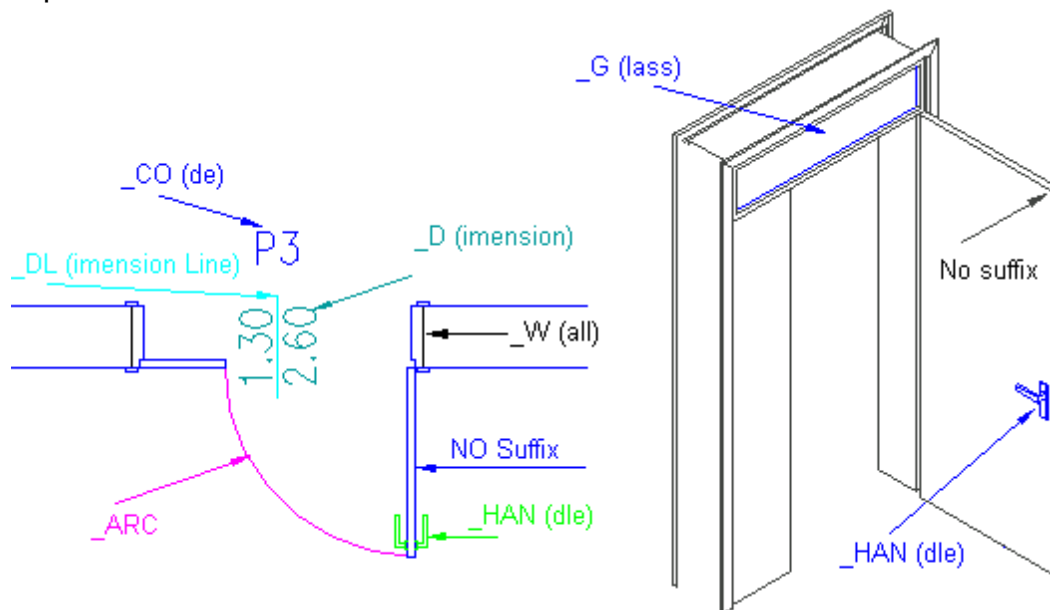
The layers of these groups of objects normally have "openings", "doors" or "windows" as the root of the name.



As far as the layers generated by the windows, those in the following figure are especially important.

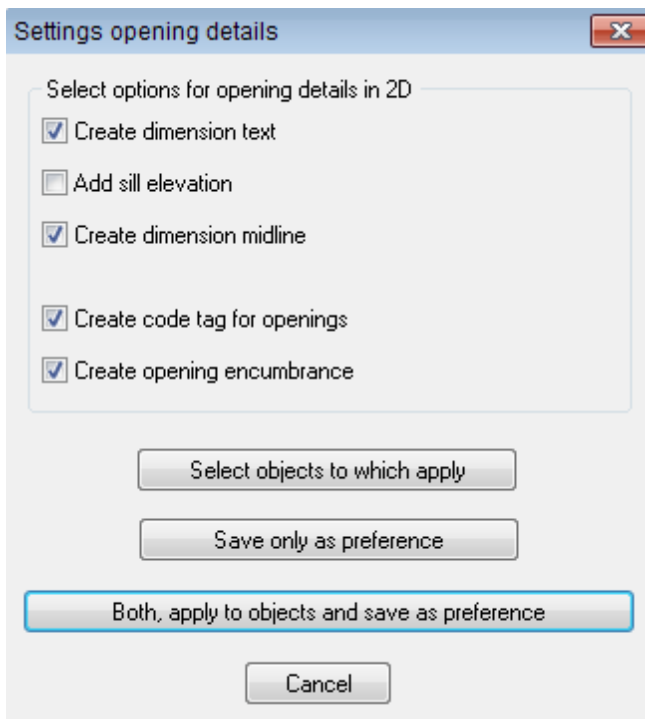
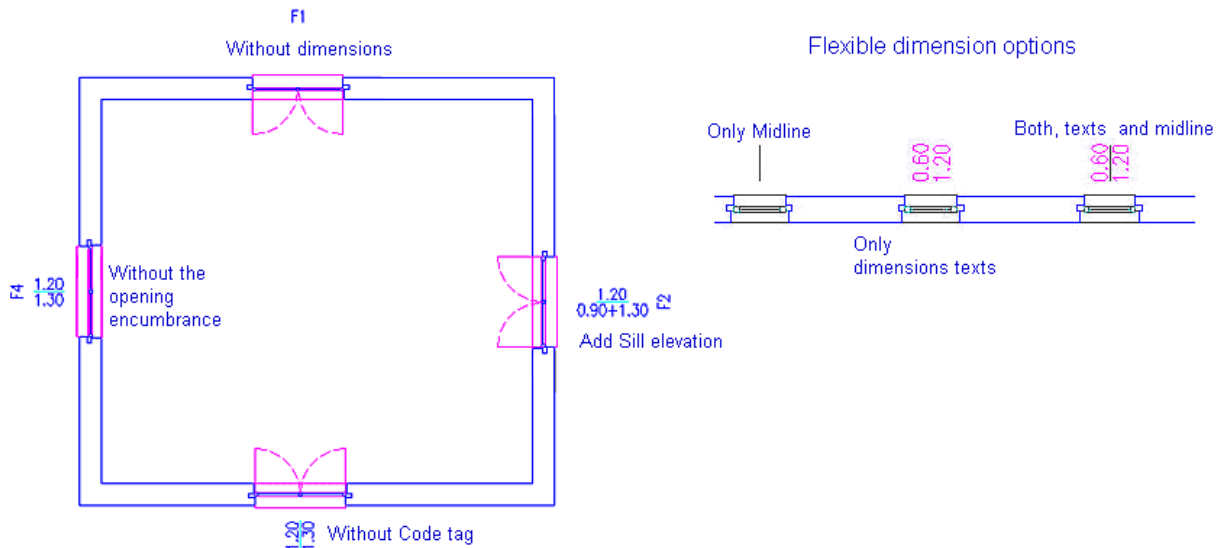


As far as the layers generated by the doors, those in the following figure are especially important.



Opening drawing options

It is possible to optionally draw and view a series of details concerning openings. Midline, dimension text, window opening encumbrance and more. Each detail is placed on different layers so that you can set a color at will, the type of line and the viewing state. The OPENSET command shown introduces an even greater degree of flexibility. The following figure illustrates the dialog box with the various options and drawing possibilities of the same openings.



The dialog box of the command allows you to operate in two ways.

The *Select objects to which apply* button allows you to select as many openings as you wish and to modify them based on the options selected.

The *Save only as preference* button allows you to choose the settings to be used for the subsequent insertions. If you wish AddCAD to always act in a certain way, for any new product, just memorize the desired options in model drawing.

The *Both, apply to objects and save as preference* button allows you to do both with just one button, to change the default settings and to modify the openings based on the new settings.

Delete openings

The AutoCAD *DELETE* command can be used with AddCAD opening objects. Aside from deleting the object, lines and arcs are closed automatically. The cancellation of the selected openings and pillars automatically restores the lines of the wall as they were before being broken. In the case of pillars, the original lines are recomposed by following the logic of the most probable configuration of the lines before the pillar was inserted. In complex situations it is impossible to recompose the lines and therefore a manual intervention with AutoCAD commands is required.

Move opening dimensions

The MOVEDIM command is very convenient whenever you wish to move the position of the opening dimension texts. For example, if for legibility reasons it is better to stretch the dimensions of a window to the inside, the command allows you to move both the entire dimension composed of texts and midline or just the text or just the line along the axis of the midline. Notice that the new positions become a property of the object. This means that if the object is modified subsequently, the new position of the dimension text does not change. Once the dimension element of an opening has been positioned properly, the command allows you to apply the new position to other selected objects.

Command: MOVEDIM

Enter to move only the text or [All/Text/Line]:

Select an opening:

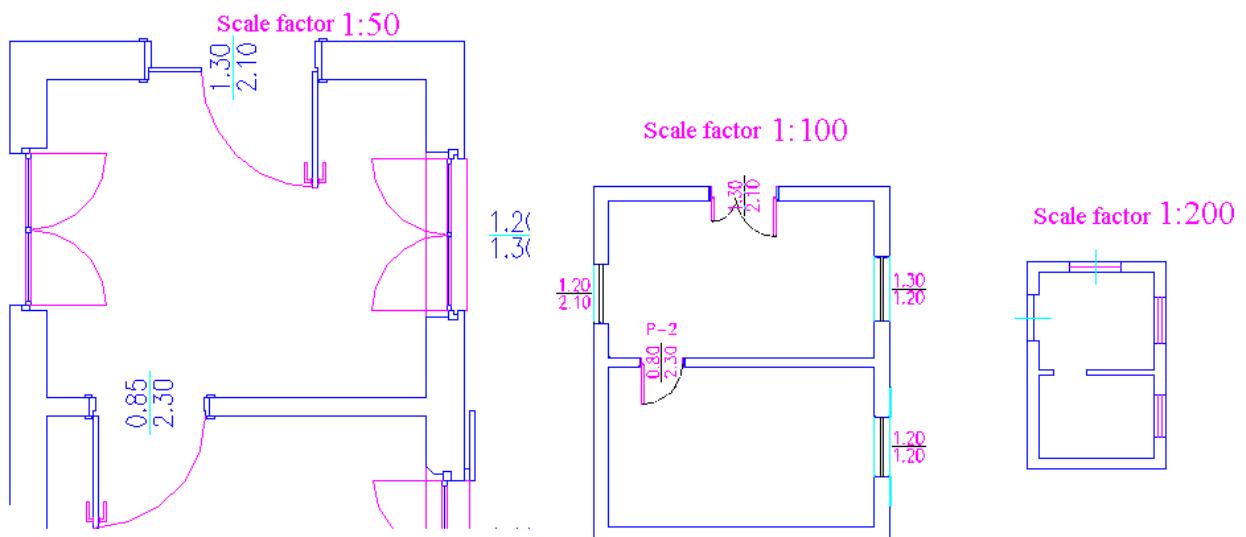
Specify distance from the inside line or [Inside]:

Select openings to which apply the same distance or Enter to exit:

The position is constrained to move in the direction of the midline axis. The origin of the distances is at the intersection between the internal line of the room and the midline axis. If the *Inside* option is activated, it is possible to provide the value of the distance towards the inside. It is normally enough to indicate a point on the drawing by referring to the dragging image which appears on the screen.

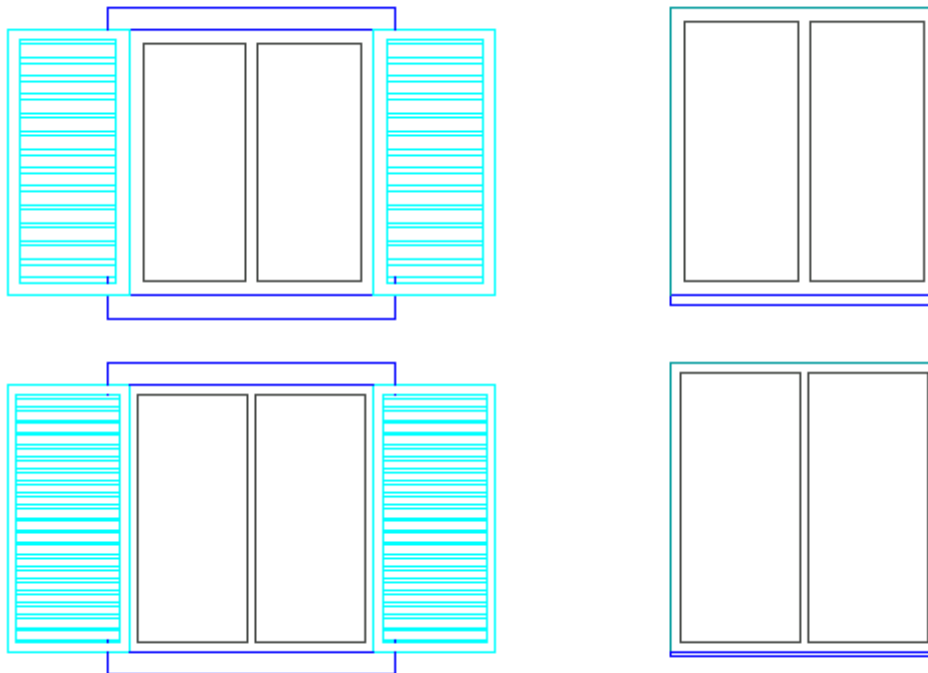
Use of several scale factors

Openings, windows and doors generally have three scale representations: detailed, generic and cadastral. The first is generated for scale factors below 90, the second from 91 to 199 and the third from 200 on. AddCAD has a [command which allows you to regenerate one of the three representations](#) possible by simply changing the current scale factor.



The dimensions of the text changes along with the scale factor. As the scale factor varies, the drawing of the openings is generally different. The representation with a scale factor 1:100 has magnified details and many details are even canceled (see the lines of the frame). With the scale factor 1:200, the representation is that of the cadastral plans. An enlargement of details similar to 2D is also done in the 3D model.

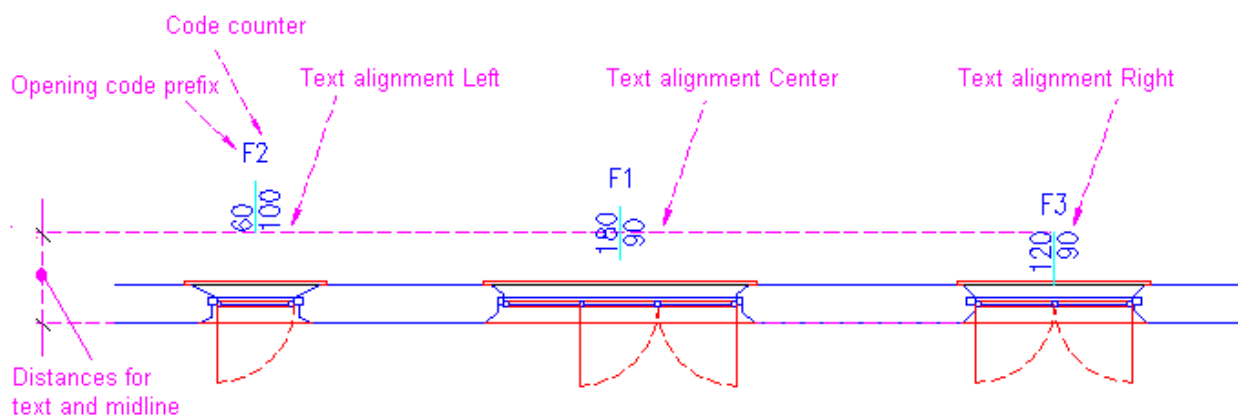
Scale factor 1:100



Scale factor 1:50

Position and text styles of dimensions

It is possible to choose how to align the texts and lines of the dimensions. The length of the line and the height of the text depend on the scale factor and therefore can be configured with the [SCALEF command](#).



Floors and Views generation		Openings and walls 2D/3D	
Dimensions		Parametric and stair layer	Quantities survey system
Windows Midline distance: <input type="text" value="0.7"/> Dimension text distance: <input type="text" value="0.7"/> Text alignment: <input type="radio"/> Left <input checked="" type="radio"/> Center <input type="radio"/> Right		Dimensions Minimum value: <input type="text" value="0"/> Length of extension lines: <input type="text" value="0.3"/> Move if less than: <input type="text" value="0.3"/> Displacement: <input type="text" value="0.25"/> Total dimension offset: <input type="text" value="0.35"/> <input type="checkbox"/> DIN dimension <input type="checkbox"/> Comma as decimal separator <input type="checkbox"/> Round to 2,5mm	
Doors Midline distance: <input type="text" value="0.05"/> Dimension text distance: <input type="text" value="0.05"/> Text alignment: <input type="radio"/> Left <input checked="" type="radio"/> Center <input type="radio"/> Right		Text style for openings dimensions: <input type="text" value="ADDCCDIMST"/> Decimal places: <input type="text" value="2"/>	

The position of the opening dimensions can be configured in the *Dimensions* tab of *AddCAD* options. You will notice that it is possible to define your own text style for the dimensions and to set the number of decimal

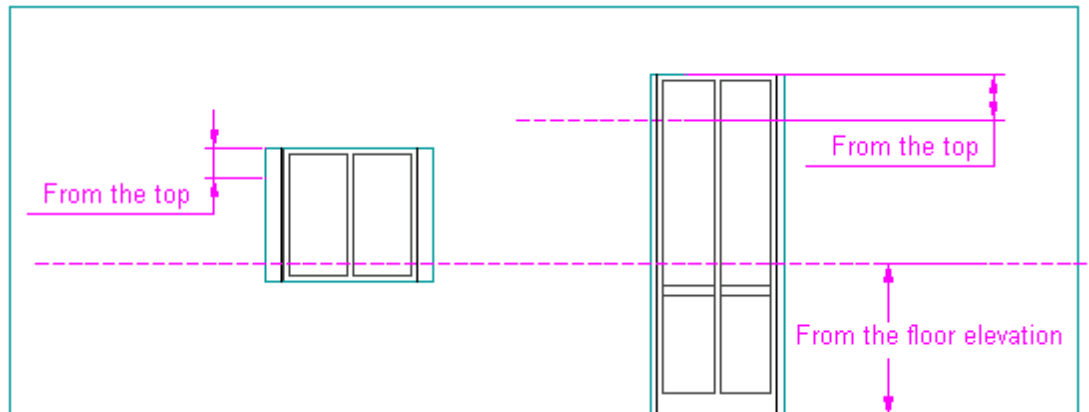
places. Furthermore two sets of data have been adopted, one for windows and the other for doors. Even though there are actually door objects which use window data. In general the door settings are used when the 2D graphic of the object allows you to draw the dimensions on the wall level, without creating illegible situations.

Last of all you will see that once the openings have been inserted with the set position, they cannot be modified with the [MODIFY command](#). The MODIFY command maintains its original setting. The only way to change the position of the dimensions is the [MOVEDIM command](#).

Lighting and plaster deduction

Lighting surfaces

Openings with glass frames and windows are parametric objects considered lighting and therefore they have a surface which is considered lighting. The amount depends on the structure of the object. The surfaces, typically width*height, of the windows are considered lighting surfaces. As far as the height is concerned, you can intervene in the Lighting deduction boxes of [Quantities survey system Tab](#) of *AddCAD* options, to take into account the presence of parapets, balconies, eaves, etc. and therefore to deduct height from the lighting surface.



We can see in the figure that for windows, the real deduction regarding floor elevation is equal to the distance between the floor elevation and the windowsill dimension, if positive, otherwise it is zero. As far as the distance from the top, the deduction is always net because it starts from the highest dimension of the window.

Plaster deduction surfaces

All the openings, including doors and windows, have surfaces which are detracted by the quantities survey system of the plaster and of the wall. If the area of the opening is greater than the edit box *Don't deduct if less than* of the [Quantities survey system Tab](#) of AddCAD options, then plaster deductions take into account the difference between the area of the opening and the area specified in the box *Amount not deductible* in the same dialog box.

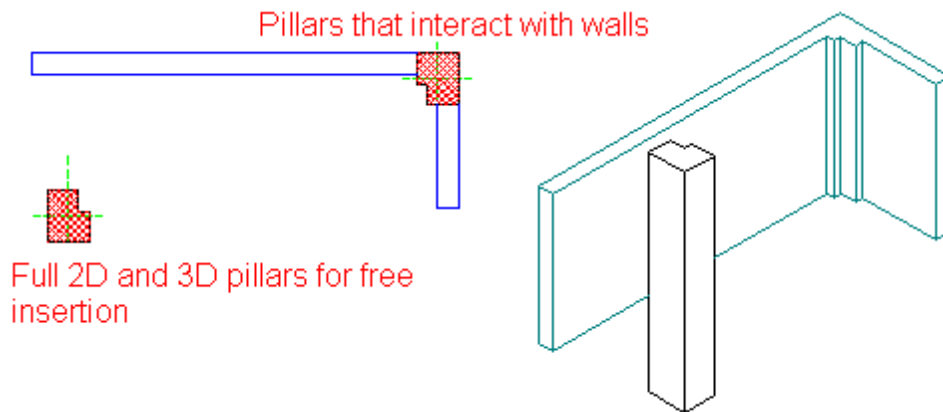
Pillars in distribution layout

AddCAD has two groups of pillars. From a 2D viewpoint, they are identical. However they have 3D characteristics and interaction with different walls.

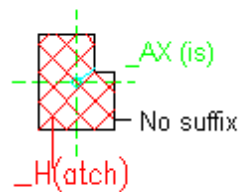


The first group contains pillars inserted in walls, partially or fully overlapping them. These objects have plenty of features. They break wall lines in order to have a correct 2D graphical representation and their 3D representation is generated automatically at floor height, together with the 3D wall generation, properly taking into consideration the polygon of the contour of the pillar itself. Furthermore, like for normal openings, if they are canceled, the 2D wall is automatically closed.

The second group also forms a graphical standpoint, are not bound by walls, they have their own parameterized 3D representation and can be used when the above-mentioned automations are not wanted.



Pillar layers



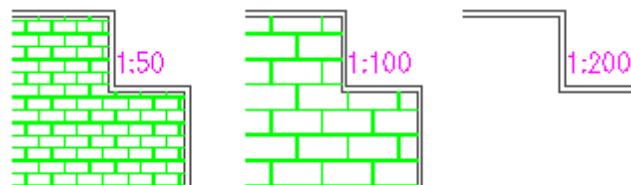
The suffixes used in the names of the pillar layers are shown in the figure. The hatch scale of the pillar varies depending on the current plot scale factor and can still be modified by means of a specific parameter requested to the user upon insertion.

Balconies

This group also includes elements for drawing balconies which can be fully parameterized. They are both balconies complete with slabs, and railings and parapets to be inserted on slabs.

2D and 3D representation

Balconies always have a two-dimensional representation and a more complete three-dimensional representation.



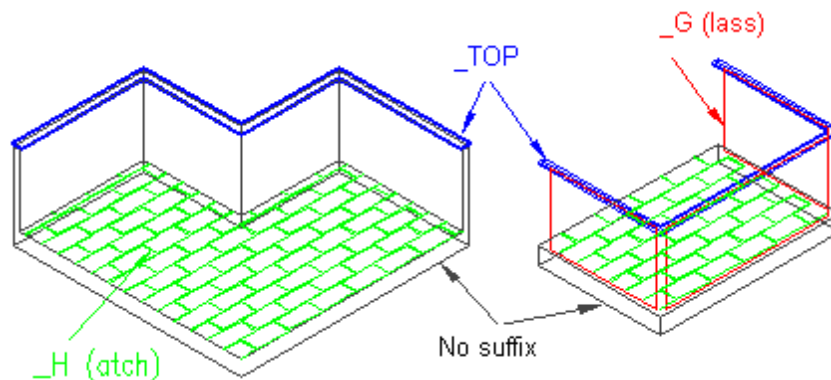
Hatches and scale factors

Balconies have a hatch on the *2dbalconies_h* layer and a Z dimension floor elevation. Actually they are important entities even with a 3D view. The size of the hatch depends on the parameter assigned when the object was inserted and the current scale factor. In the sense that for cadastral plans, the hatch is not generated at all, whereas for lower scale factors the size of the hatch depends on the scale factor.

Note on Z Elevation parameter

Balconies are inserted automatically so that the treading surface corresponds to the

current floor dimension at the time of insertion. For this reason we recommend, in many cases, leaving *Z Elevation* at zero. If you wish to raise or lower the balcony respect to the floor surface, answer with a value greater or less than zero.



Balcony layers

The suffixes of the layers created for the insertion of balconies are illustrated in the figure. Obviously the root is "balconies".

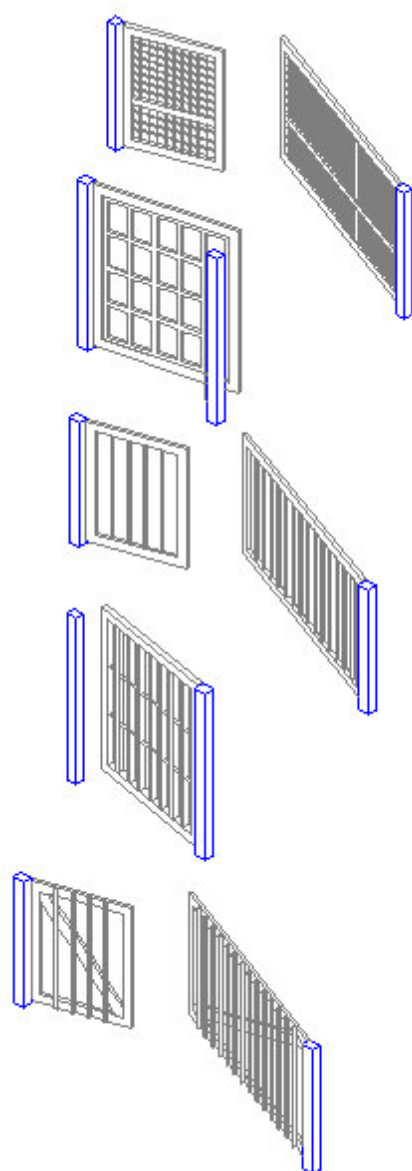
Gates and 3d trees

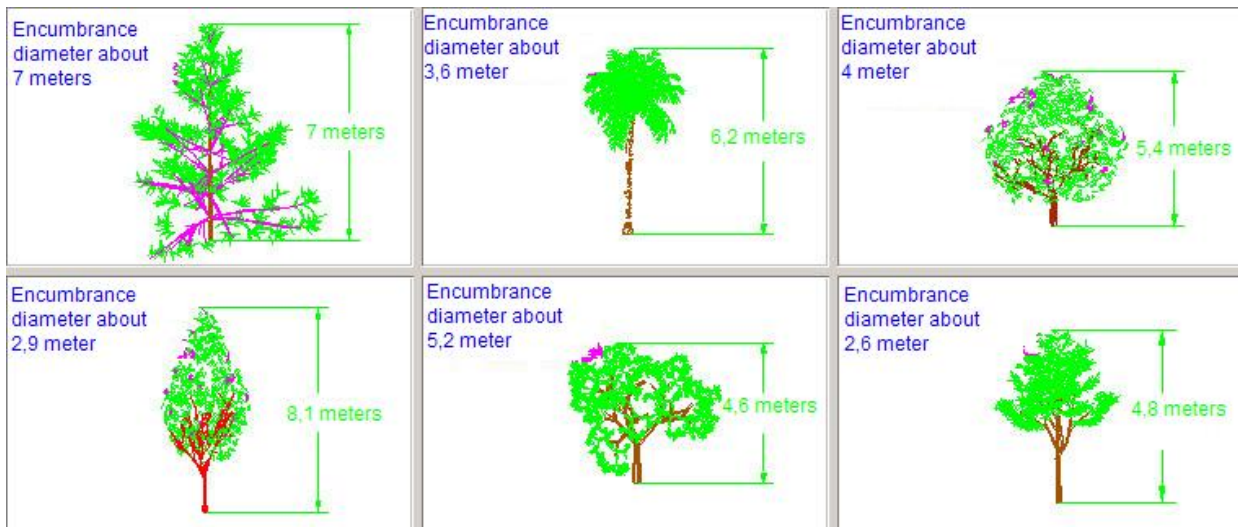
Among the external accessories, the gate series is very flexible.

In some cases, you may choose whether or not to insert a crossbeam. The opening degree of all gates is changeable. This allows you to draw them in variable positions. It is possible to dimension the height, the size of pillars and holes, the number of bars and the insertion dimension.

Gate width

The *Overall width* of the gate and the *Width on the left side* have a particular function. Besides allowing you to choose the size of the single leafs as you please, it makes it possible to generate a gate with just one leaf opening on the right or left. If *Width on the left side* is assigned a zero value, then the gate will be built from just one element opening on the left side. If on the other hand both are assigned the same value, then the gate will have one element opening on the right side.

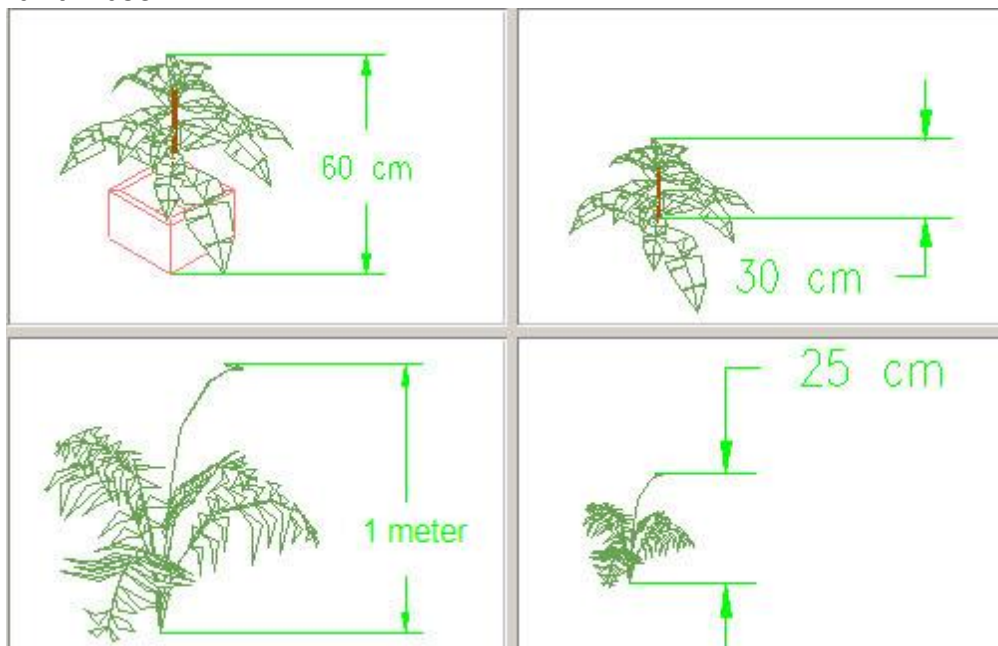




The library also has a group of trees and a group of plants in 3D. These have limited parameterization capabilities.

We've still made it possible to be able to change the names of the layers. In practice, the layers of trunks, leaves and branches can be modified when the objects are inserted.

The images of the selection menu show the plan and height dimensions of the various trees. Obviously if they are inserted in drawings which work with different units of measurement, the objects are inserted automatically with the dimensions relative to the unit in use.



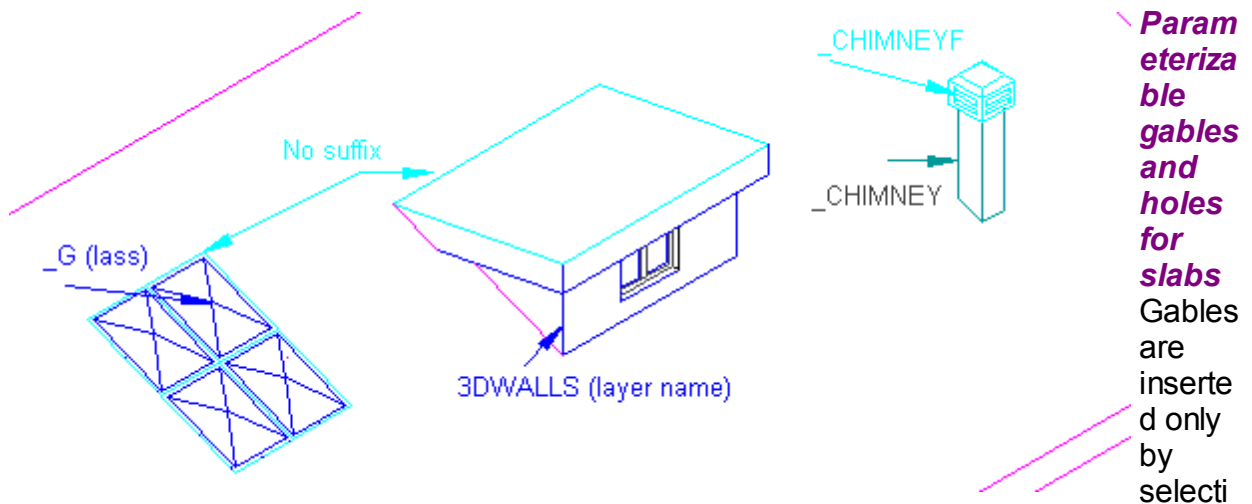
Roof elements

This group of objects contains some types of roofs and slabs, gables and skylights,

chimneys, openings in windows for gables, and curved roof elements.

Roofs and slabs

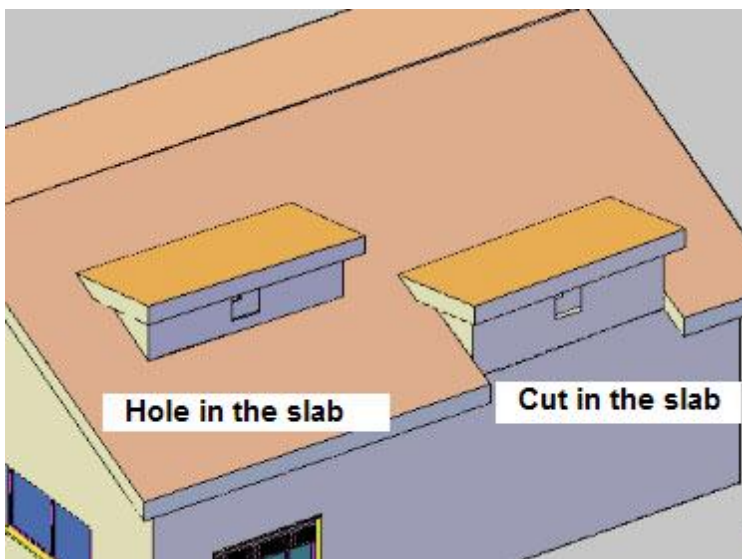
For drawing complex roofs and coverings, the [roof creation and edit commands](#) are used. For simple roofs upon which you do not want to add other elements and where there is no attic distribution to be built, some objects represent a quick alternative to the roof drawing commands.



ing an inclined slab. The slabs which can be selected are all those created with the roof creation commands. The insertion of these objects is preceded by the request:

Select the slab on which to insert the object:

Once you have selected the slab, just position the gable at the desired point and angle. It can help to choose the favorite fixed point. Just make sure that the encumbrance shape of the gable is included in the slab before typing the insertion point. The insertion of a gable automatically causes regeneration of the slab, automatically creating a hole or cut where the gable is positioned and modifying any existing hatch.



Cut in the slab

The *Cut in the slab* parameter allows you to decide whether to make a cut or create a hole on the slab.

Parameterizable holes

The same considerations hold true for parameterizable holes. Holes can be created on slabs by selecting contours with the [roof modification command](#). The parametric holes have been left due to compatibility with the older AddCAD versions. Horizontal partitions created with the [FSLAB command](#) can also be selected for

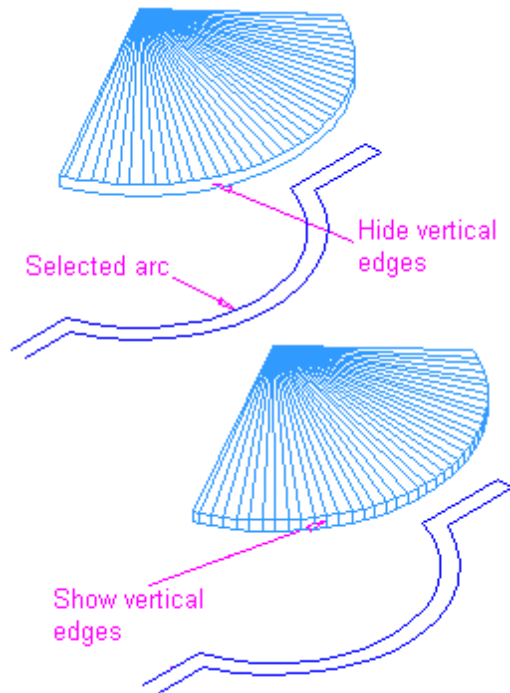
inserting holes.

Gable windows

These objects are normal windows, without 2D representation and without automatic insertion functions, to be inserted in gable openings. The insertion is very easy if you notice that the insertion point is on the left angle, inside the opening and the point on the gable can be indicated with the object snap. The angle can always be assigned by means of the object snap by selecting the right angle inside the opening.

Chimneys

As for gables, you must select the roof element on which to attach the object. The chimney will be perfectly aligned with the roof if it is placed inside the selected slab.



"roofs".

Roofs on curved wall

You are requested to select the arc of the internal face of the curved wall. The roof consists in a cone section which has as (x, y) coordinates of the vertex, the center of the arc selected. The start and finish angles of the cone correspond to the start and finish angles of the arc of the external wall face. If there is no jump (slope), as for a slab, the cone sector degenerates in circle sector. The curved roof is approximated by flat faces. The number and amplitude of the wall faces depends on the *Approximation of curved elements* variable of the [scale factors dialog box](#). We recommend inserting the openings on the curved wall, generating the 3D curved wall and generating curved roofs, again with the same approximation value.

Layers of roof objects

The suffixes of the most common layers created with the insertion of roofs are shown in the figure above. The root of the layer name is typically

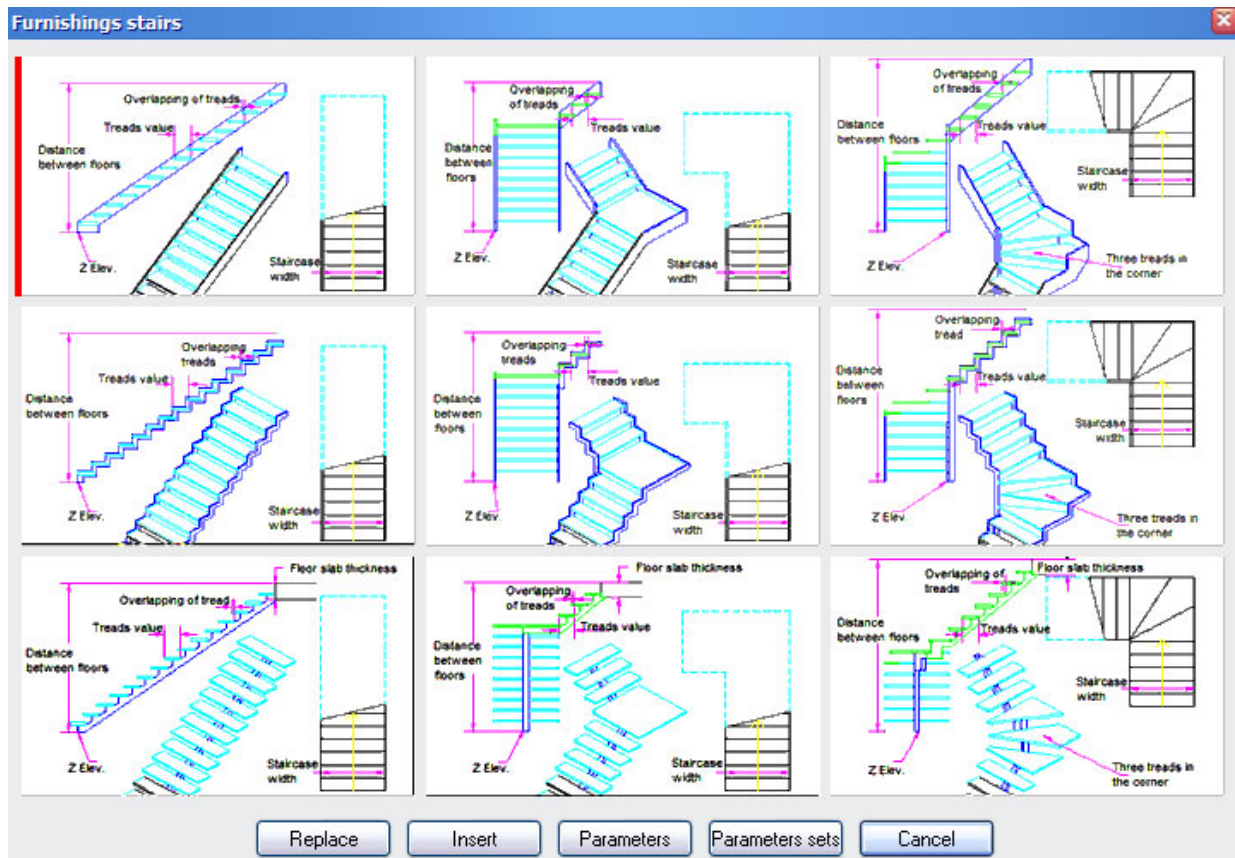
Erasing roof elements

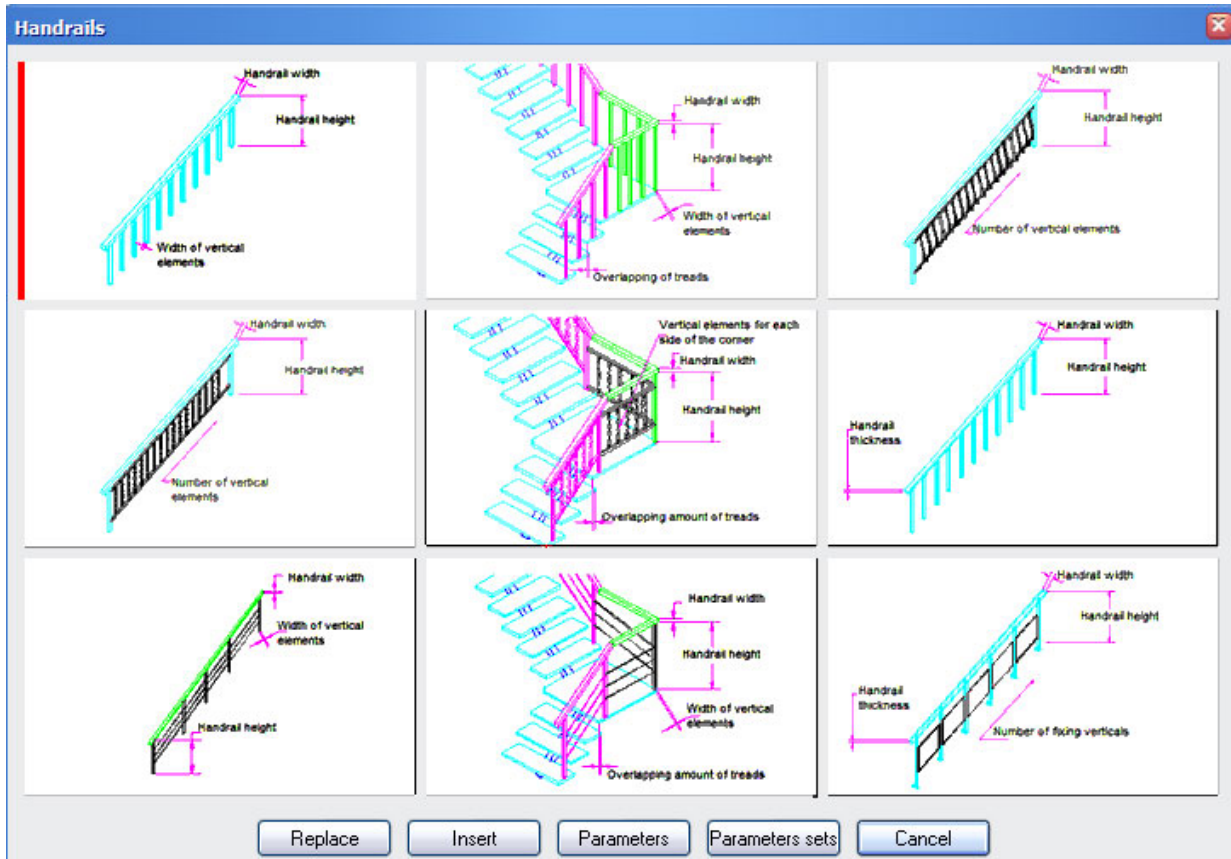
The AutoCAD DELETE command removes the selected roof object and closes the hole on the slab made when it was inserted. If for example we insert a gable on a slab, AddCAD automatically generates opening of the gable. If we decide to delete the gable, the underlying hole is also closed. If there is a hatch, it is regenerated as well.

Furnishing stairs

A series of furnishing stairs is included in AddCAD's parametric library. These furnishing stairs are used inside for the most part. They can be divided in three different groups, each of which is distinguished by its bearing structure. The figure below shows the selection box of the stair model. Each group is made up of one linear staircase and two angled staircases. Of the two angled staircases, one has a simple landing in the corner

and the other has the possibility of having two or more treads in the corner; this variant is to be chosen with the specific parameter. Among the various parameters, you can see the presence of *Overlapping amount of treads*, typical for inside furnishing stairs.



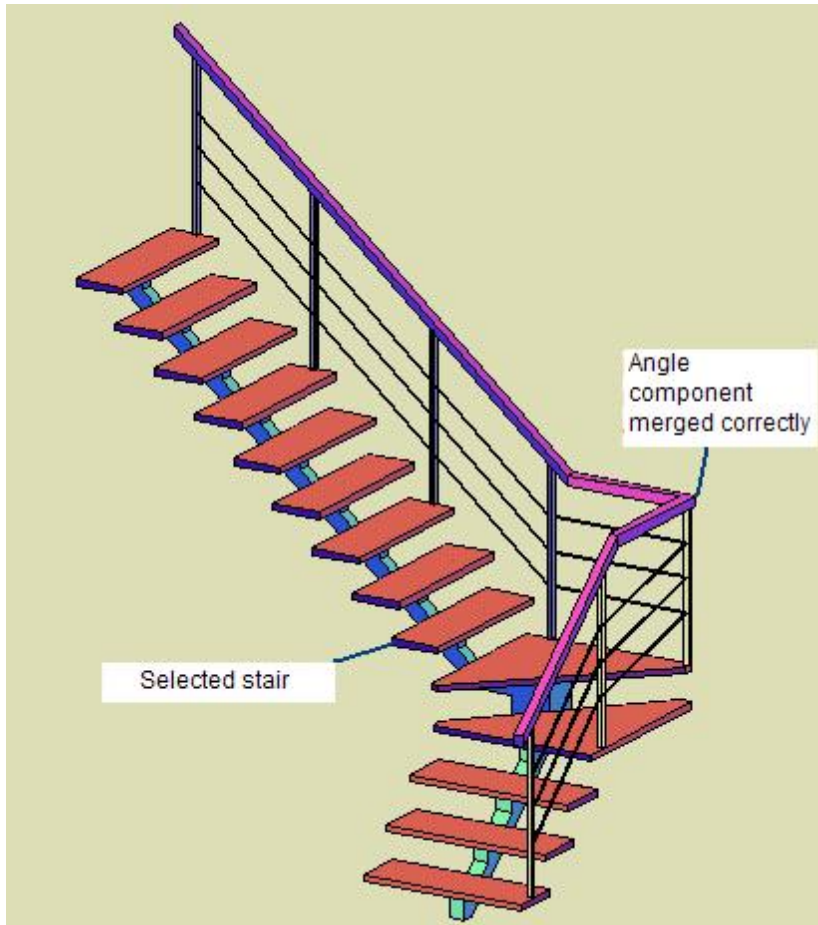


The wire type is a type of handrail which is used very much on indoor staircases.

One type of object is part of AddCAD's parametric system, merged angle handrailing components.

As can be seen in the figure below, by simply selecting the stairs, a perfectly merged angle component is inserted with the other linear elements.

These new models can also be used with angled staircases present in the [staircase library](#).

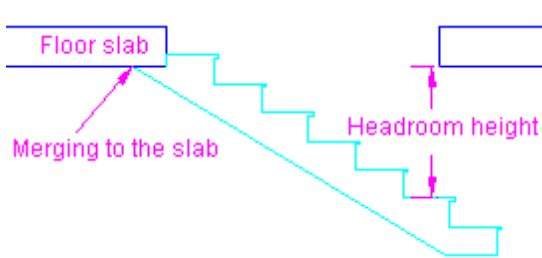


Parametric stairs

When the stairs are simple enough and do not require a double 2D plan representation for the starting floor and ending floor, the parametric stairs are a suitable alternative to the [stair modeler](#). They need no definition model and therefore are definitely quicker to draw. There are different types of parametric staircases; spiral staircases and staircases with single or multiple flights. Some are already supplied with handrail, while others are set up to be attached to one on the concerned flight side. The handrail also has a parametric alternative to the continuous handrail, which are parametric handrails for individual sections. The [HANDRAIL command](#) can also be used for parametric stairs and therefore you are given the option of making a choice fit for the individual case.

Distance between floors or riser

There are two different ways of acquiring the construction parameters: assigning a specific value for the riser or assigning the distance between floors to walk on the stairs. In this second case, the program calculates the value of each riser based on the number of risers. The type of parameterization depends on AddCAD options. In [Parametric and Stair layer tab](#) there is a control box *Inter floor distance for parametric staircases* which allows you to switch from one method to the other.

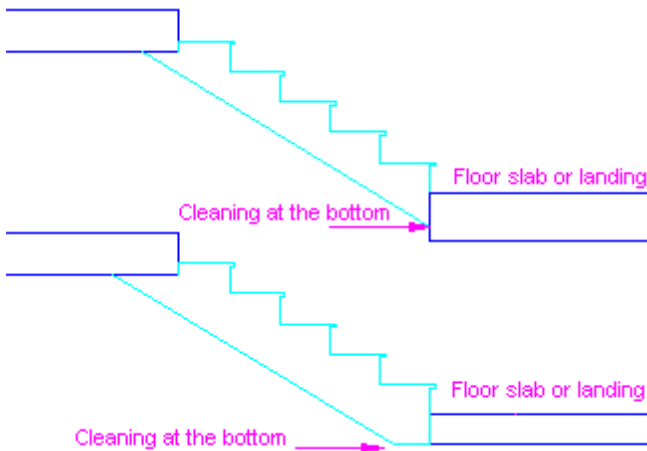


Last riser and merging to floor slab

Staircases with the *Floor slab thickness* parameter include this thickness in the distance between floors. Furthermore the final riser is represented by part of the floor slab thickness. In reality, the object is built with one less riser as it is assigned as a parameter. It merges automatically with the upper floor slab.

Parameter Headroom height

In stairs with this parameter, an opening can be made in the floor slab which provides headroom at a specific height. Obviously the stairs must begin from such a height that the *Distance between floors* is greater than the *Headroom height*.



Merging with the landings and floor slabs downstairs

Single flight stairs can be merged with landings and floor slabs. In fact the presence of parameters *Z Elevation* and *Floor slab thickness downstairs* are a useful tool to attach the staircase to a floor slab or starting landing at a certain height. When the single flight starts from ground level, the two parameters are placed at zero. The shape which the bottom merging takes on depends on the parameters. You can select *Z Elevation* directly from the drawing by indicating a

point with the object snap.

Insertion of handrails

After having parameterized the handrail in its constructive dimensions, the insertion of parametric handrails continues with the following message.

Select the stair on the side where to insert the handrail:

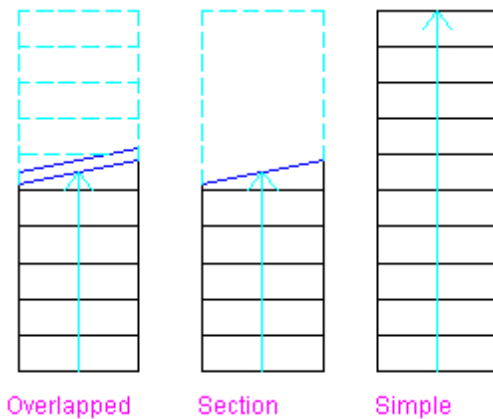
You must select the staircase on the appropriate side.

Parameter Number of approximation elements

The parameter *Number of approximation elements*, present in slides and curved steps, indicates the degree of approximation of the curved element. The use of a surface model requires you to approximate curved figures with a finite number of flat elements. For most applications, an approximation of about 15 elements per right angle is sufficient. If you wish to increase the curved effect, you can increase the number of approximation elements.

Inverting direction in multi-flight and spiral staircases

Spiral staircases and some staircases with more than one flight of stairs go in one direction. To be able to invert direction, just mirror them.



Representative models of plan staircases, 2DSTAIR command

Staircases have 3D and 2D representations. 2D stair representations have three graphical versions depending on whether the stairs merge the floor of the plan drawing with the downstairs floor, with the upstairs floor or if there are both. The section lines are placed at the point where the stairs cross one and a half meters. If the flight of stairs begins above this dimension or ends below it, the section lines will not be visible. When the staircase is inserted, the model generated is the one chosen in [Parametric and Stair layer tab](#) of AddCAD options. Once the

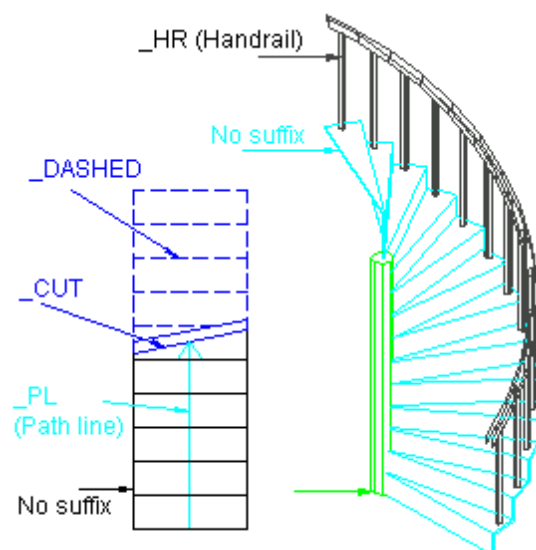
stairs are generated with a certain representation, they can be changed with the 2DSTAIR command.

Command: 2DSTAIR

Stair 2D representation[Overlapped/Section/Simple]<Simple>:

Select the stairs to which change the 2D representation:

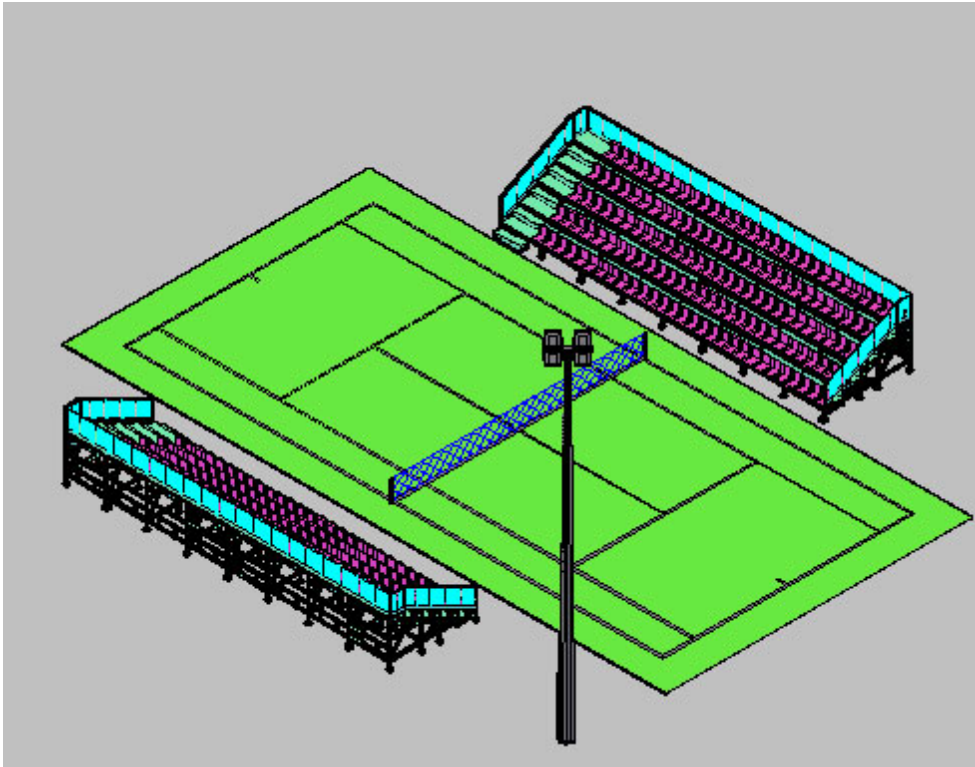
You must indicate the type of representation and select the stairs for which to change the representation model.



Stair layers

The suffixes used most often are shown in the figure.

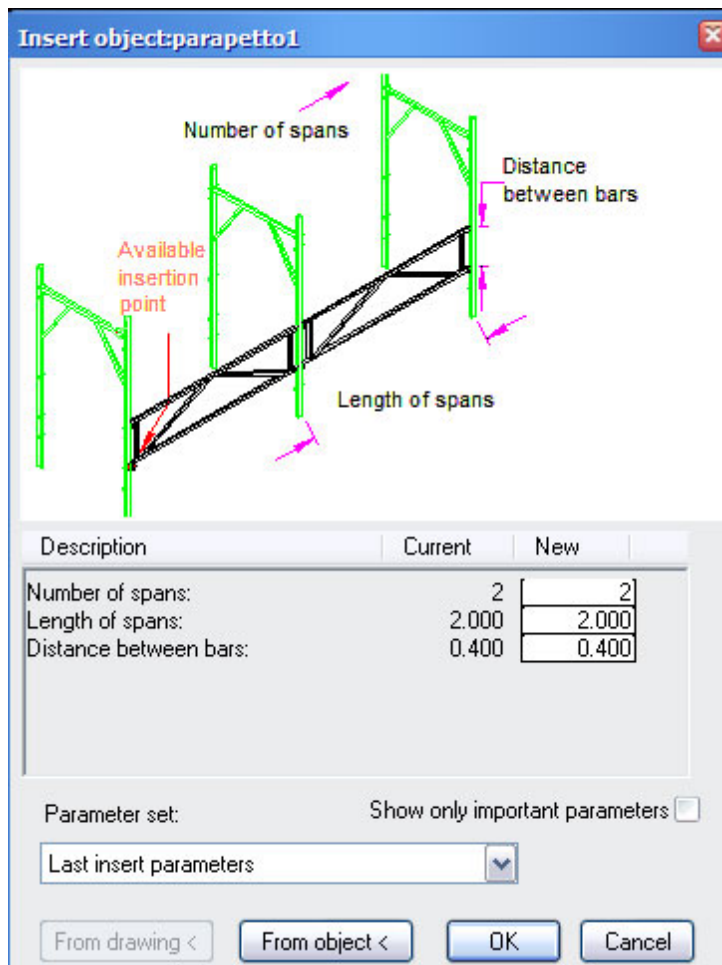
Sports facilities objects



AddCAD includes a series of parametric objects which help to design sports facilities. Besides playing fields such as football, basketball, tennis etc, elements such as stands, parametric steps, CONI-approved fences and light towers are in the library. The image has examples of parametric

objects for drawing sports facilities.

Objects and system for scaffolding



AddCAD includes a new system and a series of objects for making scaffolding in 3D.

The composition and drawing of the scaffolding with frames is possible by means of the required parametric elements. These elements have a special insertion technique which allows you to select important points, through which it is possible to work in axonometric projection without making insertion errors. Implementing a three-dimensional scaffolding is therefore extremely fast.

There are lower and upper bars, racks, parapets, boards and diagonals.

The parameters relative to the number and size of the spans are particularly important.

Connected elements such as diagonals and parapets have important insertion points to facilitate insertion in an axonometric viewpoint.

The possibility of using the *remarkable insertion* points greatly

speeds up building complex scaffolding.

By varying the parameter *Z Elevation* it is possible to go on with the next floors, the second floor and so on.

In the figure at the side, you can see an enlargement of the image with the parameters of a parapet which can be inserted by choosing the appropriate point on the rack.

Walls

Introduction

The three-dimensional model generated by AddCAD is a surface model, the so-called 3dface. AutoCAD users know these entities hide objects and have visible and/or invisible edges. Furthermore they are entities which can be processed by rendering programs. This type of approach has many advantages. In architecture, solid modeling applied to designing on a whole is not very feasible. The drawing of a three-dimensional wall is therefore contained completely by a set of 3dfaces, making edges invisible. Each floor is generated separately. Walls can be [generated at a height of the floor](#) or else there can be different heights for different walls. AddCAD also allows you to [generate variable height walls](#), like those below a sloped roof slab. Curved surfaces, such as curved walls and arced openings, are approximated by a series of flat faces. The approximation value can be modified in the SCALEF command dialog box. AddCAD also allows you to implement openings in curved walls, inserting an opening even complete with frame, treated by [the parametric library](#). The approximation value of the curved wall also affects the generation of these parametric objects. This is why the wall face will be correct only if the same approximation value is used both while generating the object and generating the wall.

When modifications to the layout are made, such as the modification or deletions of openings or the insertion of new walls, AddCAD is capable of automatically updating both the 2D and 3D wall representations.

The generation of wall faces is done automatically while respecting the holes for the doors and windows present in the drawing. The way in which the 3D wall representation is generated depends on two groups of options. The first group establishes options valid for the entire wall model. These options are managed by [Openings and walls 2D/3D Tab](#) of the AddCAD options dialog box. The second group of options regards individual sections of walls and faces. Different values concerning wall height and edge visibility can be set with the [DDWALL command](#). The generation of some sections of the 3D wall can also be linked to architectonic elements such as floor [slabs, roof slabs and stairs](#). It is not necessary to use these commands if you wish to generate 3D with standard settings and with top and bottom edges visible. Disabling visibility of the top and bottom edges is very useful when you wish to generate walls which will be merged to other walls on other floors without wanting to see the separation lines between floors (floor marking lines).

Constant height 3D walls

The WALL3D command generates the three-dimensional wall model of a constant height floor. The walls can have different heights. To achieve this result, you must assign height attributes to the entities of the 2D representation. The WALL3D command was specifically designed for walls with an optional number of openings and pillars. The openings can have any type of polygonal shape and as many vertices as you wish. This also allows to approximate the type of arc, circular, elliptic, etc. The only constraint is that

of using parametric objects. When you wish to use frames created as AutoCAD blocks, they must be inserted inside simple parametric openings which only define the hole contour. An alternative to parametric openings is that of boring the wall with the [WHOLE command](#). The WALL3D command automatically generates the faces of flat walls and curved walls. It must be said that one condition necessary for the three-dimensional development of walls is the presence of the 2D drawing. If this information is missing, automatic generation cannot take place.

The WALL3D command proposes the following options.

Select an object on the floor that you want generate, Enter to generate the current floor:
If you press Enter, the 3D wall model of the current floor will be generated.

Standard wall generation height

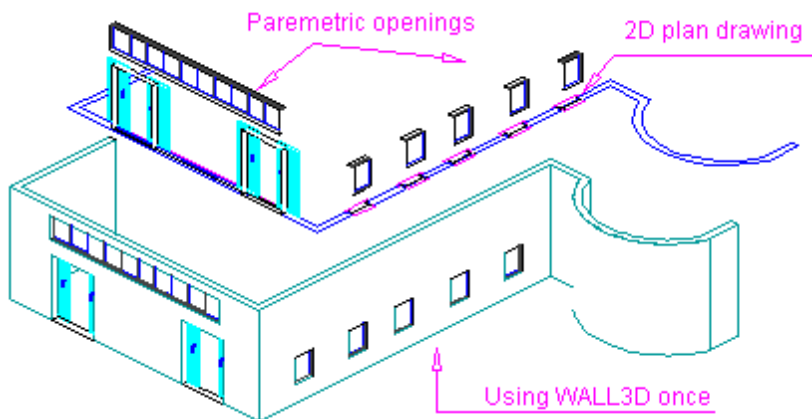
The default height with which the wall is generated depends on the floor height set for each individual floor in [Floors and views generation tab](#) of the ADDOPTIONS command.

Generation of a selected floor

If you select an object from which the floor elevation can be found, the 3D wall model of this floor will be generated.

3D surface insertion layer

AddCAD automatically inserts 3dface entities generated by WALL3D and VARW3D on the layer with the named formed by joining ADDC3DWALL\$ with the floor elevation. It is also [possible to use more than one layer name](#) for 3D wall generation. In this case, the names are obtained from the entities of the plan drawing by replacing "2D" with "3D". If for example you have lines on the ADDC2DWALLX\$3.20 layer, the 3D surfaces will be placed on ADDC3DWALLX\$3.20.



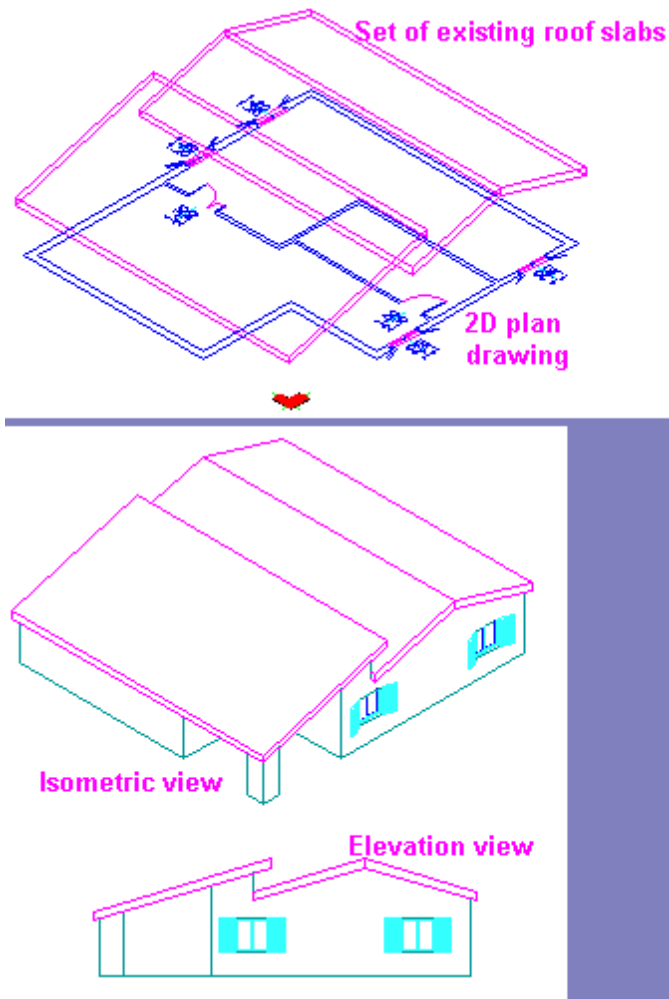
Generation of an individual façade

The W3DFACE command generates and updates the three-dimensional wall model of an individual façade for each entity selected on 2D. The command repeatedly requests to select a line or an arc.

Command: W3DFACE

Select a line or arc of the wall facade that you want generate, Enter to exit:

Variable height walls



The command allows you to generate variable height walls. This is required for example when you wish to generate the 3D wall model of attics or walls below stairs. The surfaces generated will usually consist in 3dface entities alone. The dialog with the user proceeds as follows.

Command: VARW3D

Select an object on the floor that you want generate, Enter to generate the current floor or [Alignment/Slab/One wall face]:

Generation of a selected floor

Like the WALL3D command, it is possible to select an object which does not belong to the current floor to create the 3D model on a non-current floor. If however you still choose to generate the 3D model of the whole floor, the command considers the set of slabs drawn with the commands for roof creation, the set of stairs, the distribution of walls and the openings of the floor. In fact distribution of the floor can contain windows, doors and openings at will which can be positioned freely. The encumbrance polygons of the pillars are also

considered. The zones of the plans not covered by roof slabs, floor slabs or stairs are generated at a constant height as if they were generated by the [WALL3D command](#).

One wall face

The whole roof system is always considered. It does not process the entire plan, the single faces are selected individually. Lines and arcs must be selected from the 2D plan drawing.

Slab

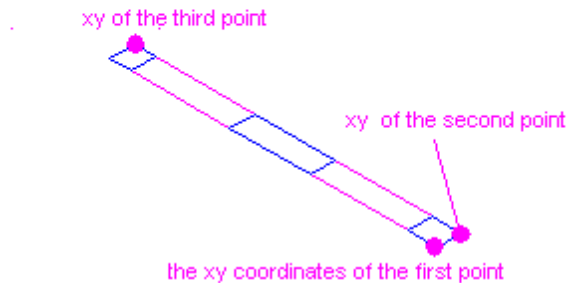
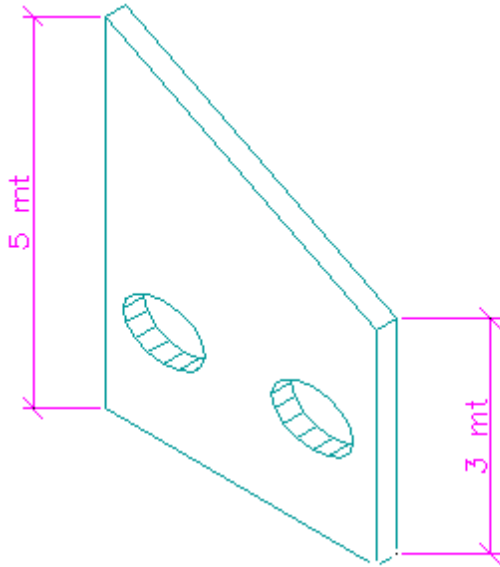
The face upon which the top edges of the faces are cast can also be defined with the individual slab. If you choose Slab, the requests are:

Select a slab:

Once you have calculated the roof slab, the command repeats the request message to select an entity belonging to the wall face you wish to generate. It can be pointed out that the parts of the lines and parts of the arcs which are not below the selected slab create wall faces at a constant height and value equal to the height of the floor.

Definition of alignment plane

This last model allows you to generate variable height walls without the presence of roof slabs. The method consists of indicating AddCAD with three unaligned points which define the inclined plane. Let us suppose that we wish to generate the wall in the figure for example. As soon as we have set up the plan elements, we start the VARW3D command. In this case, it is convenient to use the filter mechanism of the AutoCAD coordinates together with the object snap. The coordinate filter allows you to distinguish between two coincident points that are on different elevations. The operation is described in the dialog with the command line as reported below.



Specify the first point of the plane: xy

Select XY of:

Still need Z of: 3

Specify the next point of the plane or [Undo]:
xy

Select XY of:

Still need Z of: 3

Specify the next point of the plane or [Undo]:
xy

Select XY of:

Still need Z of: 5

Select a line or arc of the wall facade that you want generate, Enter to exit:

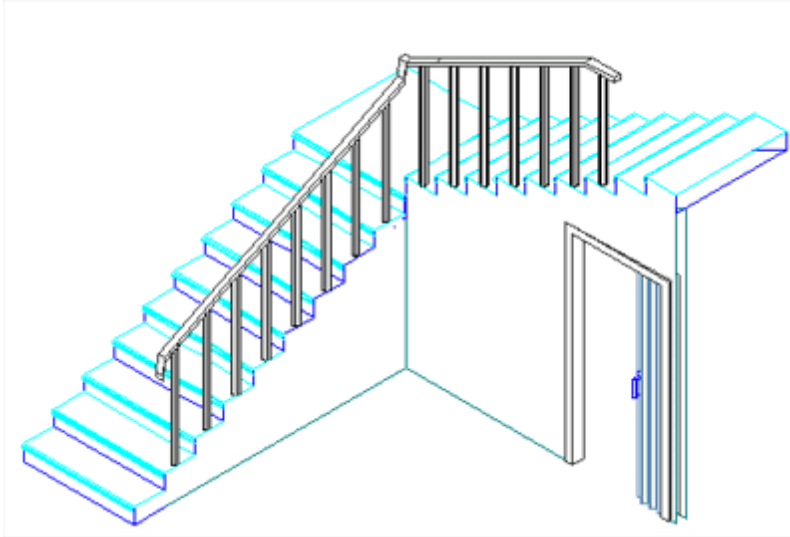
The four faces to be generated are selected.

Constant height walls even with the command generate variable height walls

It is possible to select walls and to have them generated at a constant height. Just use the [DDWALL command](#).

Ignore some roofs during generation of variable height walls

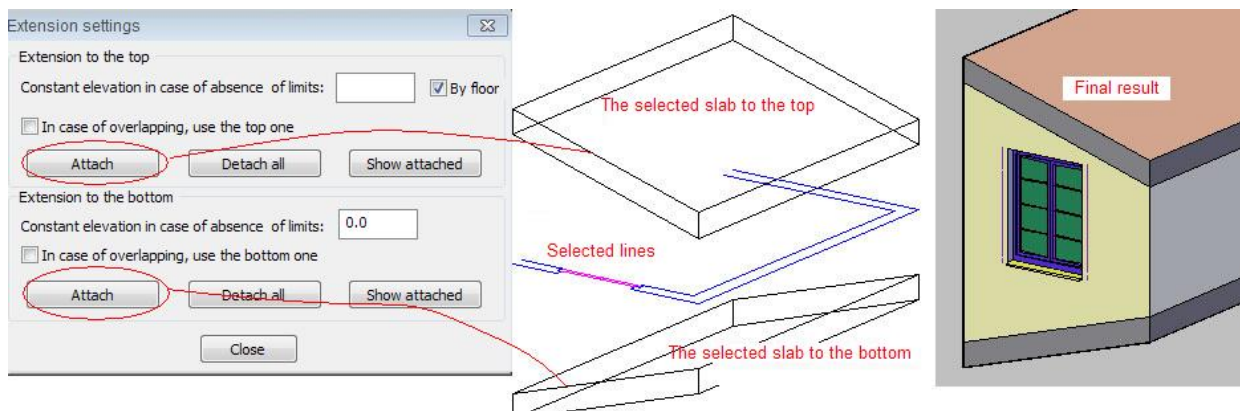
It is possible to [select a roof element](#) and to have it ignored while generating variable height walls.



3D walls attached to slabs and stairs

The information regarding automatic generation of the 3D wall representation is contained in the lines of the 2D plan layout. The command which we will describe on this page allows you to choose stairs and roof elements and to attach them to the lines of the distribution layout in order to generate their 3-D representation respecting the slopes of the soffits and other parameters of the selected components. Actually this command is not always necessary to generate variable height walls aligned to roof soffits. The [VARW3D command](#) already generates under-roof walls taking into consideration the presence of all the roofs thus easily and quickly resolving most situations. But sometimes it is necessary to assign a certain amount of slabs to the lines of the 2D wall. This occurs for example when you have overlapped roof slabs or when you wish to generate the bottom side of the wall as well as adhering to a specific slope.

The DDWLINK command allows you to attach, detach and show the 3D generation constraints of lines belonging to the floor plan layout, regardless of which floor they belong to. The following image shows the dialog box and the elements to be selected.



The dialog box is not modal, meaning it always remains active. Press Close when you wish to close the window.

It is possible to work both with the extensions to the top and to the bottom, using the top

and bottom part of the dialog box respectively. The three buttons allow you to:

- attach as many slabs as you wish to a certain number of lines of the plan layout; you are requested first to select the lines and then the group of slabs or stairs to extend
- detach all, meaning that all the merging information is canceled from the selected lines
- show the slabs and stairs attached to a line, which is requested.

When several slabs are selected and some of them overlap during generation of the wall, it is possible to specify that the bottom/top one can be used as an extension of the wall.

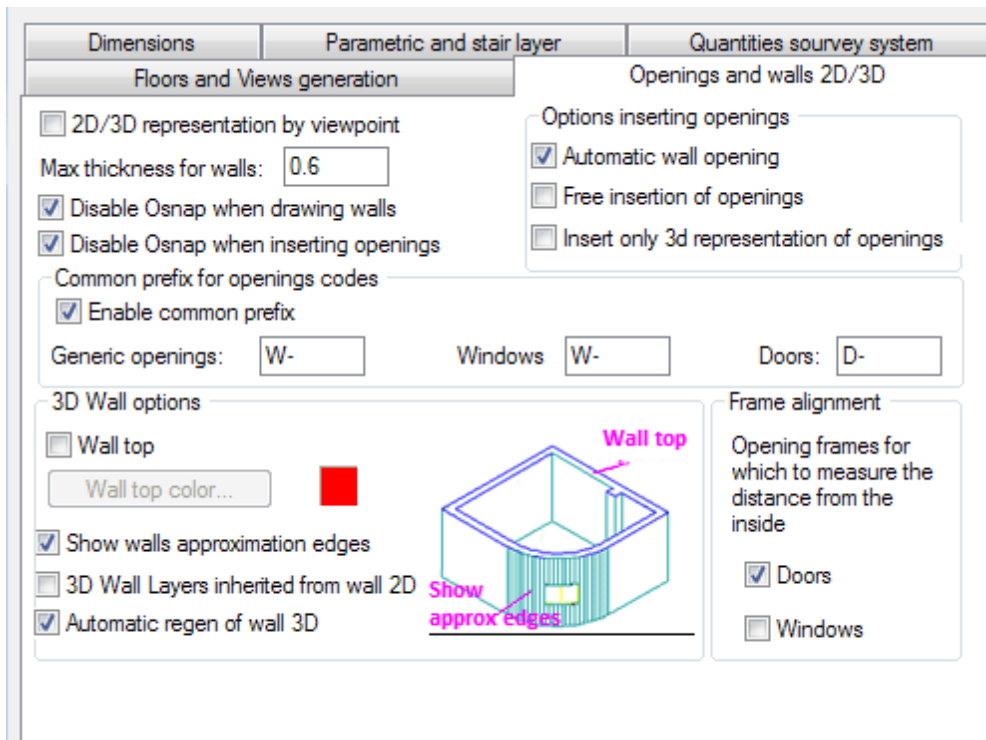
Another property regards those sections of the lines which are in zones where there are no extension objects, such as lines coming out from the roof. In this case, it is possible to specify an elevation value for the generation at a constant height. By default, the top dimension coincides with the floor height and the bottom one with floor elevation.

Assigning limits and WALL3D command

As is known, the [WALL3D command](#) generates walls at a constant height. For lines attached to stair and slab objects, the WALL3D command applies the assigned constraints and then works like the [VARW3D command](#) (3D attics).

3D wall generation options

The *Openings and walls 2D/3D Tab* of the Options dialog box contains several options regarding the drawing of the 2D distribution layout and of the 3D wall generation.



Openings insertion and wall drawing options

Automatic wall opening

If activated, plan wall lines are broken automatically after an opening is inserted.

Disable Osnap when drawing walls

Even if the object snap is activated in AutoCAD, it is deactivated

temporarily when drawing walls.

Disable Osnap when inserting openings

Even if the object snap is activated in AutoCAD, it is deactivated temporarily when inserting openings in the wall.

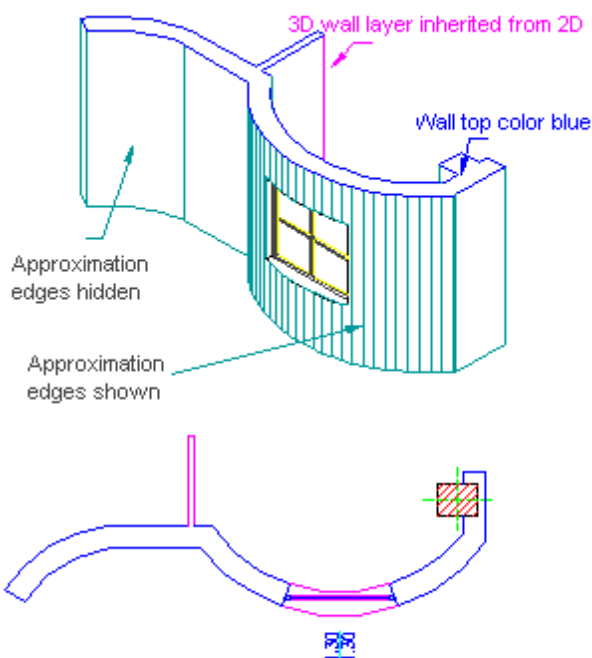
Insert only 3d representation of openings

Option which is usually activated when inserting [overlapping openings](#).

3D wall drawing options

Wall top

It is possible to activate generation of the wall top. Practically a surface is generated which closes the top section of the wall. This function slows down the algorithm and is useful only if you wish to obtain a cutaway of the three-dimensional model. The wall top should be deactivated when assembling several floors. The wall top of 3D walls only works if the distribution on a certain floor is closed. If you activate the wall top option, you may also choose its color.



Show walls approximation edges

It is possible to generate the curved wall model showing vertical approximation edges or not.

3D Wall Layers inherited from wall 2D

The 3D wall surfaces can be inserted on different layers. The root of the name is read automatically by the layer name of the entities of the 2D wall. This makes it possible to assign different materials to different walls during the rendering stage, discussed later. If this option remains disabled, just one layer is used for 3D wall generation with the root name "3DWALLS".

Automatic regen of wall 3D

The 3d wall representation can be updated automatically ensuring modifications to openings and frames. If this option is not enabled, the 3D wall model must be regenerated manually whenever necessary. By activating it, updating after each modification of holes including new insertions and cancellations of existing holes becomes automatic. Obviously it is only updated if a 3D model was generated previously. The program saves the type of generation performed in the lines of the 2D layout.

Frame alignment

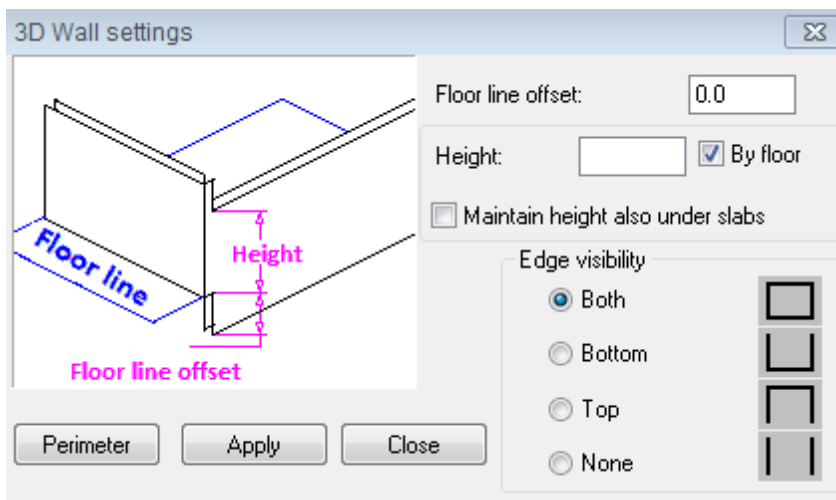
This option is aimed at providing the program with greater flexibility, specially when inserting openings in walls. Not everyone finds the solution of measuring the frame for doors from the inside of the room and windows from the outside as always being suitable. The work mode can be customized by using these options.

Keep in mind that changing the options within the same drawing leaves the option by which previous openings were inserted unchanged. This means that each opening

memorizes the option with which it was inserted and proposes the relative value when the object is modified.

3D wall settings

The DDWALL command allows you to assign the selected line and arc entities with dimensions and features other than the standard ones. In order to do this, just write the values in the dialog box and press *Apply*. The same command allows you to view the generation features of the façades. To view the generation properties, just select the lines and arcs of the 2D wall. The dialog box is not modal, it always remains active. Its values are modified based on the line and arc properties selected. Press *Close* when you wish to close the window.



Height and floor line offset

Normally the bottom edge of the face is placed on the floor elevation of the floor. The height of the façades corresponds to the height of the floor. This data comes directly from the floor list in the *AddCAD options dialog box*. Aside from this standard data, it is possible to assign both the bottom edge dimension and the top edge dimension by assigning

these parameters to 2D entities. The walls can be assigned with a certain height and this height can also be respected by the variable height wall generation command. As shown, the *Height* field has the option *By floor* which is checked to remove preset face height information and to be generated at height based on the floor it belongs to.

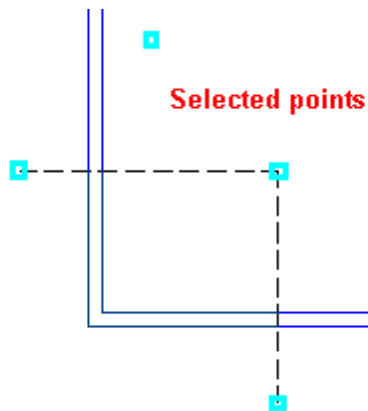
Edge visibility

You may select to make the top or bottom edges visible. These options are useful to avoid generating floor marking lines.

External perimeter of building or internal perimeter of room

It is often convenient to select all the lines and arcs of the external perimeter or internal perimeter of a room by selecting just one entity belonging to it. The *Perimeter* button allows you to select the external perimeter of a plan or the internal perimeter of a room. Without closing the dialog box, the program requests to select a line or arc of the perimeter.

Breaking lines and arcs



The BLA command allows you to break lines and arcs at intersections with a breakage indicated by a series of points.

This command is useful to quickly assign 3D wall setting options with the [DDWALL](#) command. Especially when the same faces need to have different heights.

The command asks you to indicate the breaking points.

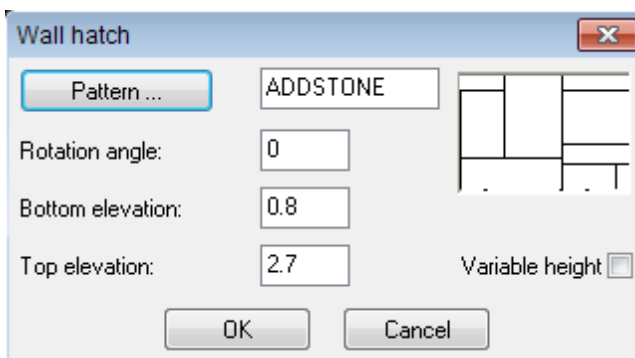
Command: BLA

First point of the breaking sequence:

[Undo]Point number 2:

....

Wall façade hatching



The HWALL command automatically hatches the façade of a wall respecting the openings, doors and windows which should be present on it. The command request to select one of the entities which identifies the façade. When there is no breakage due to frames, the façade is just one line or arc. When on the other hand frames create breakages, then the façade is made up of several lines or arcs. The

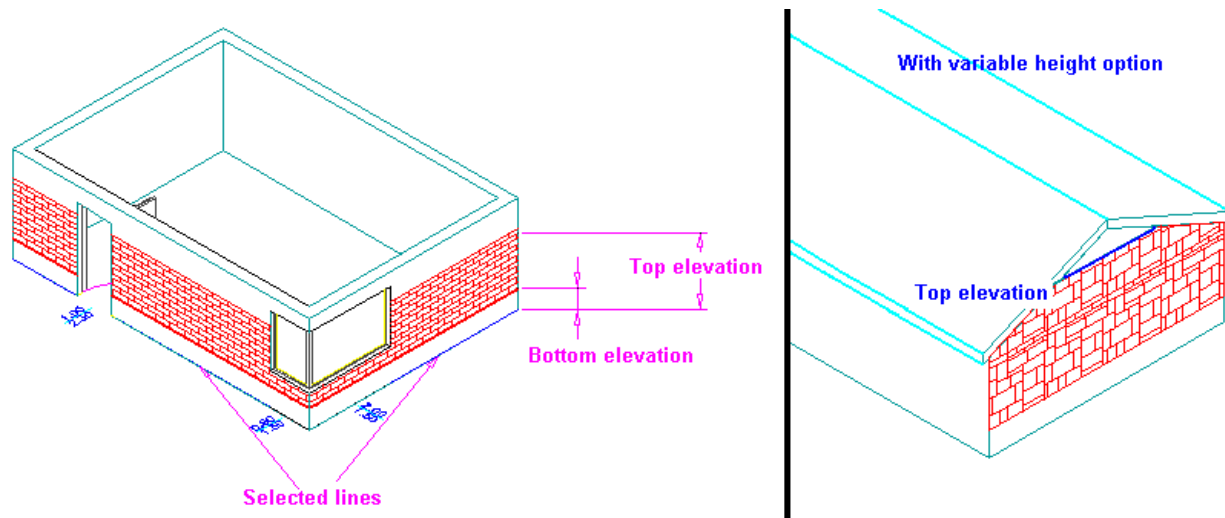
command however requires the selection of just one entity to recognize the entire façade. The command provides the creation of hatch options through a dialog box. It is possible to choose the hatch model and the rotation angle. The scale factor is modified with the [SCALEF](#) command.

If the *Variable height* option is activated, it makes the hatch on the wall follow the line of the soffit of the various slabs present, should there be roofs in the drawing. The top limit of the hatch still remains fixed at *Top elevation*. If you wish to hatch the entire wall below the roof slabs, the *Top elevation* value should be higher than the summit of the roof. The request of the command regards a line or an arc of the plan distribution.

Command: HWALL

Select a line or arc of the wall face to hatch:

The hatch is placed on the ADDC3DWALLS_H\$<FE> layer where FE is the floor elevation of the selected line. Therefore the hatch floor is found from the selected line. The hatch model library has been enhanced with new hatches. In particular for wall hatching, there is a STONE model for hatching wall façades.



Parapets

The PARAPET command allows you to trace multiline elements, namely consecutive curved and linear sections, and to modify the parameters of individual sections, or even selecting an existing parapet. It is also possible to indicate various linear sections of the parapet by simply indicating the points inside the drawing.

Command: *PARAPET*

Select a line or an arc in the start point or select an existing parapet/Enter for specify just points:

If you press *Enter* the command continues requesting points:

Specify first point:

[Undo]Point number 2:

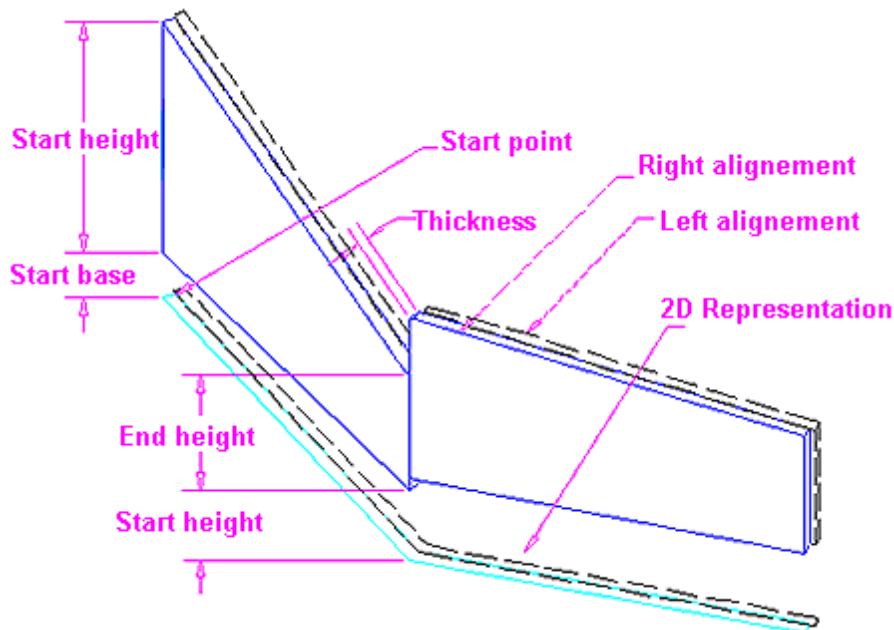
...

You can also select a parapet to modify the parameters linked to the various elements making it up. The line or arc of the entity sequence must be selected in order to immediately set the direction for the thickness. If an observer goes to the start of the line or arc and looks towards the end, the thickness is generated on the left unless otherwise

specified in the dialog box.

Layer name

The root of the layer name generated by the command can be changed. The command adds the prefix "3D" for the three-dimensional part and the prefix "2D" for the plan part. The parapet block is placed on the indicated layer unless you wish to link it to the floor. In this case, it is linked to the floor logic as usual.



2D representation

A plan representation of the parapet is generated.

Related to floor elevation

In this case, the basic data will refer to the current floor elevation and the visibility is linked to the floor with the usual layer name mechanism.

Next

Go to the next definition section. The definition sections must represent an open chain of interconnected

arcs and lines.

Save

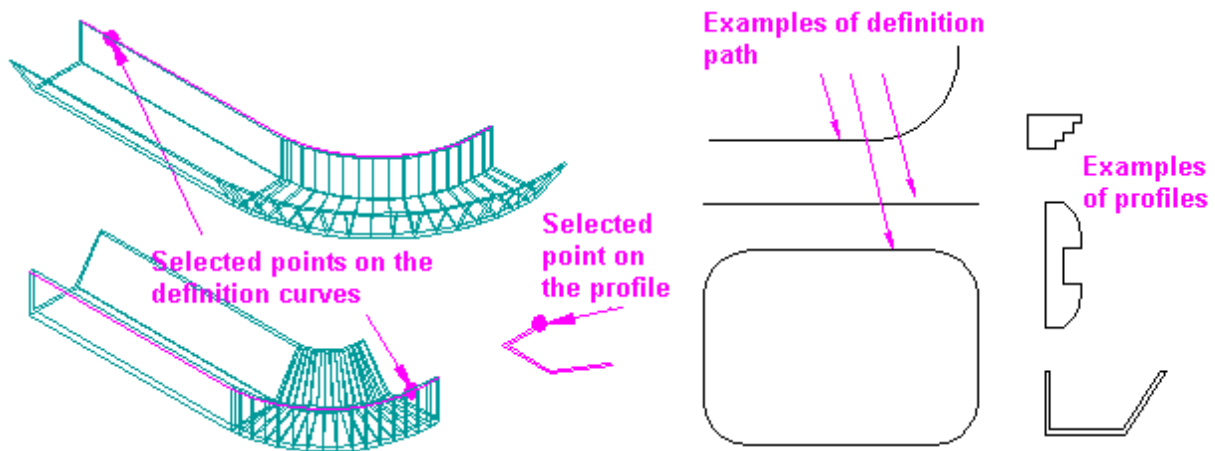
It does not generate the parapet but memorizes the data assigned to the lines and arcs in the drawing.

Create

It generates the parapet with the assigned parameters.

Drawing eaves

Many details which required a lot of processing for the solid modeller can be implemented with this command, making an easy way creating surface models. This feature meets the requirements of embellishing buildings with eaves and windowsills.



You only need to select a definition curve and a profile and the program generates the set of surfaces approximating the curved sections in the profile and in the leader with small flat surfaces. A definition curve is composed of a set of interconnected line and arc entities which can create an open or closed figure. A profile is also composed of connected lines and arcs which however always create a closed figure. The definition curve can also be indicated with points on the drawing equivalent to a set of lines with the convenience of directly selecting the correct coordinates to generate the eaves. A final method to determine a definition curve is that of selecting the external perimeter of a building. This makes it possible to apply eaves directly on the definition curve taken from the selected building.

Two aspects need to be made clear. The first regards the reference point of the profile which must coincide with the definition curve and the second regarding the direction the profile is applied on the leader.

The program requests to select the definition curve at a specific point called origin point and, for the profile, it requests to select the point on the profile which must correspond to the definition curve.

The Examples of profiles diagram shows the results of the command.

Command: GPROF

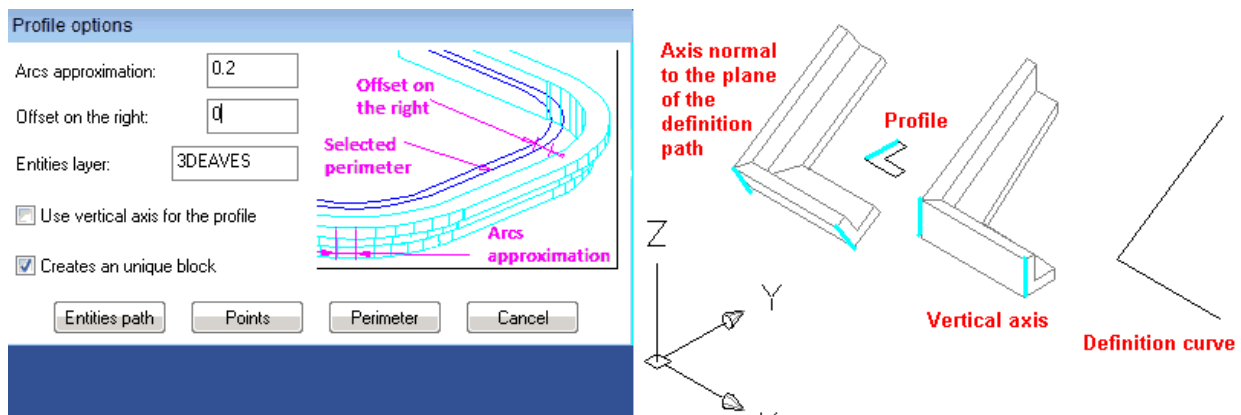
Select an entity of the definition curve, near the origin point:

Select the profile:

Specify the point that corresponds to the definition curve:

The GPROF command actually begins with a dialog box where you can choose how to define the curve and other attributes for generation of the eaves.

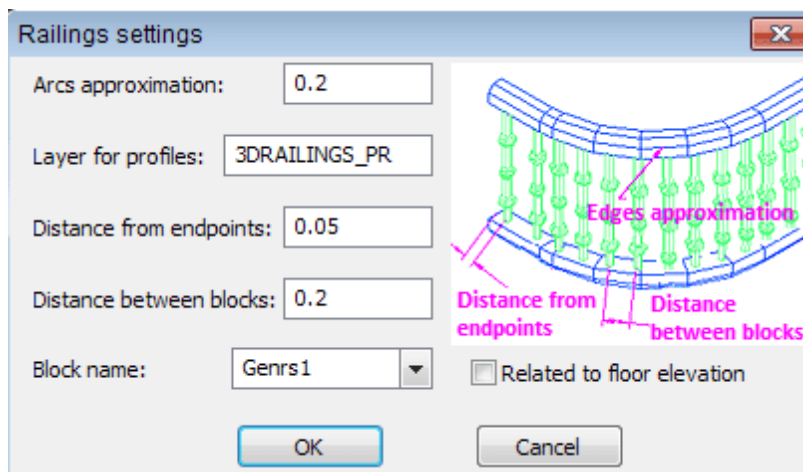
You can choose the *Entities* layer name and whether *Creates an unique block* for the entities or to individually generate all the surfaces. When there are no special requirements, we recommend creating a unique block in order to treat the eaves as one object to be able to move it etc. It is possible to move the path of the profile from the definition curve by a value equal to that assigned as *Offset* on the right and to assign a value for *approximating the curved section* of the definition curve. The approximation of the curved sections of the profile can be different since its value is set with the SCALEF command. The option *Use vertical axis* for the profile regards those situations in which the definition curve does not form a plane parallel with the XY plane. In this case, the three-dimensional development of the profile can be implemented still observing the vertical axis of the profile with respect to the UCS global reference system or not. An example shows this.



Generating railings and balustrades

It is possible to create railings, balustrades and handrails selecting any definition path in space. A definition path can be a polyline or a connected group of lines and arcs, not necessarily coplanar.

The railing is defined by an optional amount of profiles and any type of block inserted several times along the definition path.



The command for creating railings and balustrades is GENRFR. It requests some options in a dialog box.

Arcs approximation

The curved profiles will be approximated by surfaces as wide as the set parameter.

Layer for profiles

The surfaces of the profiles will be generated on the indicated layer.

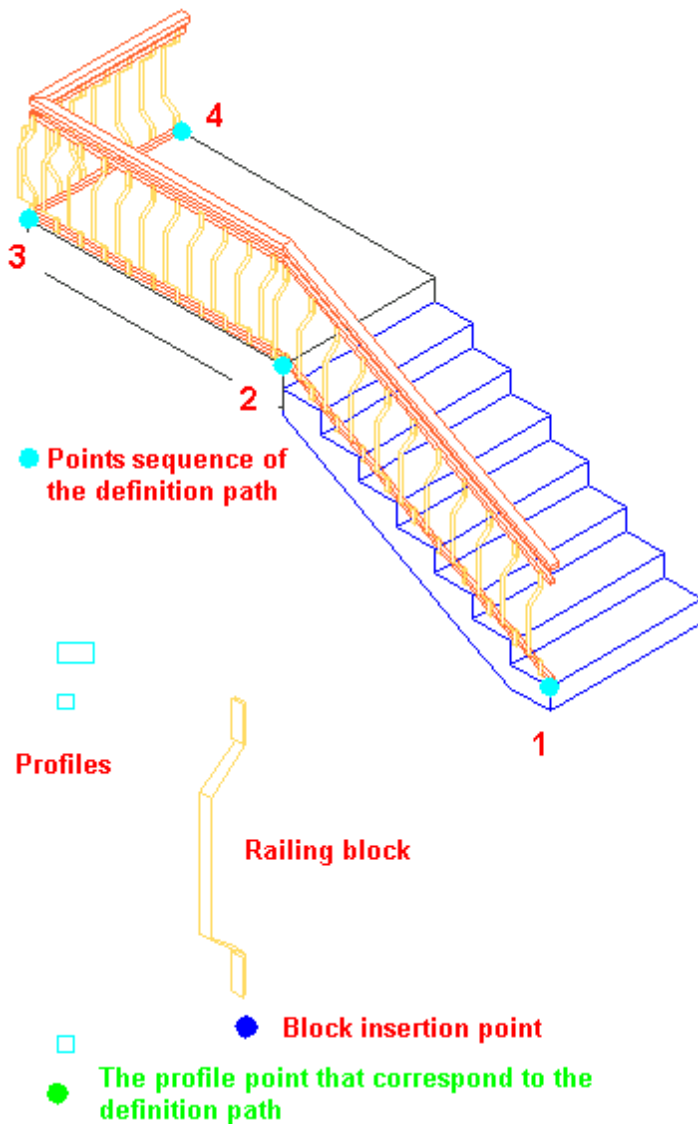
Distance from endpoints

This is the value of the

distance between the first block and the endpoints of the definition path.

Distance between blocks

This indicates the distance between two blocks. The real distance could be approximated to this value as much as possible. This depends on the length and centerline of the blocks.



Block name

The block is repeated along the indicated definition path. The list contains all the blocks defined in the drawing. You can insert a block created in another drawing and to use it in the project. The AddCAD program contains a series of blocks with relative profiles. From the menu *AddCAD Draw>3D Models->Blocks for railings*, you may access an image menu where you may select the model you require.

Related to floor elevation

All the blocks repeated along the definition path and the profile surfaces are combined in a unique block. If this option is enabled, the entire railing block will be placed on one layer. Its name will be obtained by joining the profile entity layer name with the floor elevation. This allows you to manage the railings on the floors of the construction. If this option is not enabled, the block layer will be the current layer.

Having pressed OK and exited, the command will continue to request further information required to build the object. You must define the definition path and an optional number of profiles. The definition

path can be indicated by a series of points, or by selecting either a closed or open polyline or a similar contour made of lines and arcs. The profiles, which are also polylines or line and arc contours, must be closed.

The order of the points of the definition path and the point where the entity belonging to the definition path is selected is important to properly insert the blocks and profiles.

Command: *GENRFR*

Specify path start point or [Select entity]: **Select**

Select an entity of the definition path, near the origin point: **<selection of closed or open definition path>**

OR

[Undo]Point number 2:

[Undo]Point number 3:

..... at the end, select the first point if you want a closed definition path

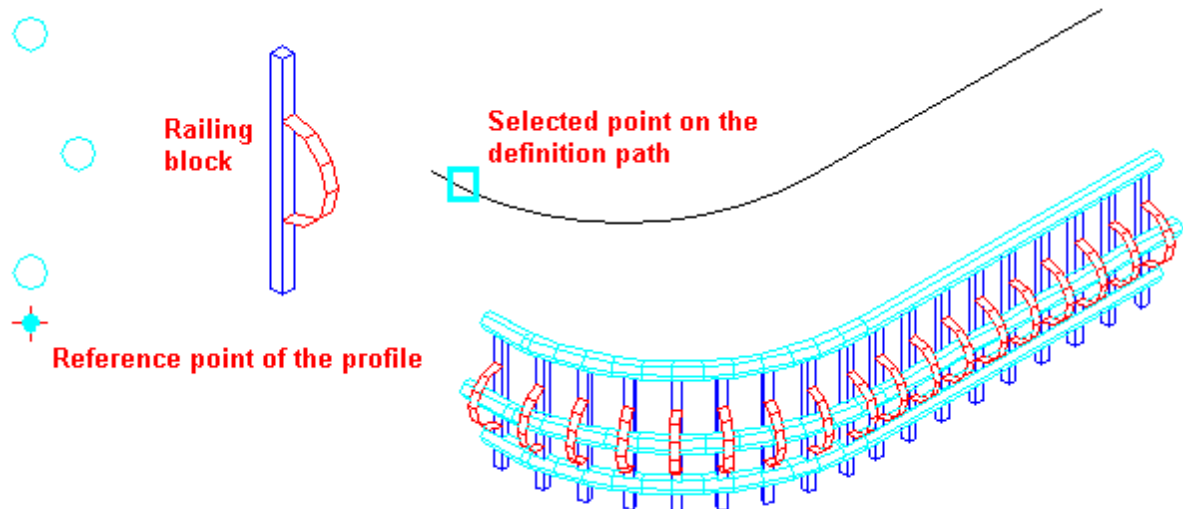
Select the first profile or Enter for none:'

Select next profile or Enter to exit:

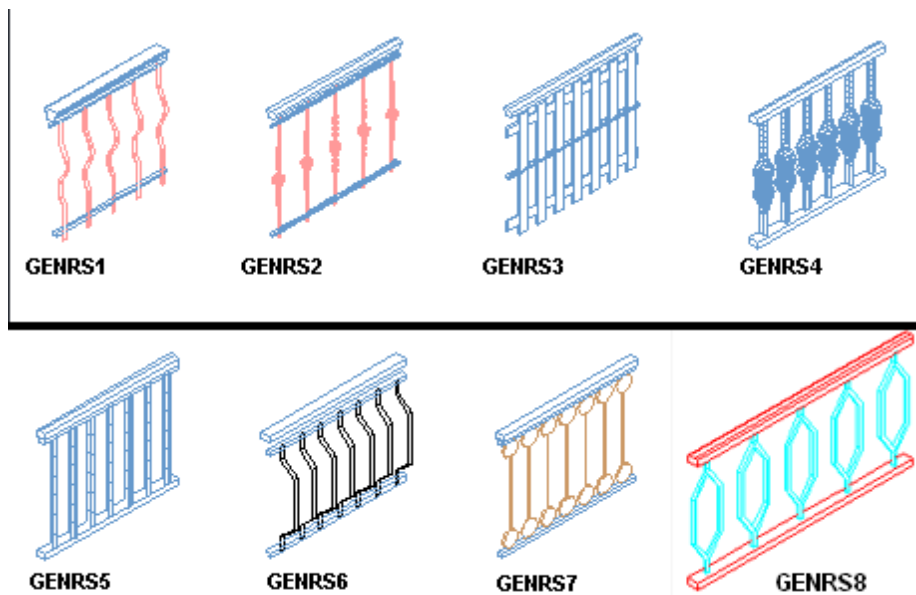
.....

Corresponding point of the profiles related to the path line:

This point is important to define the position of the profiles with respect to the definition path. The images of the two examples on this page explain how to use this command.

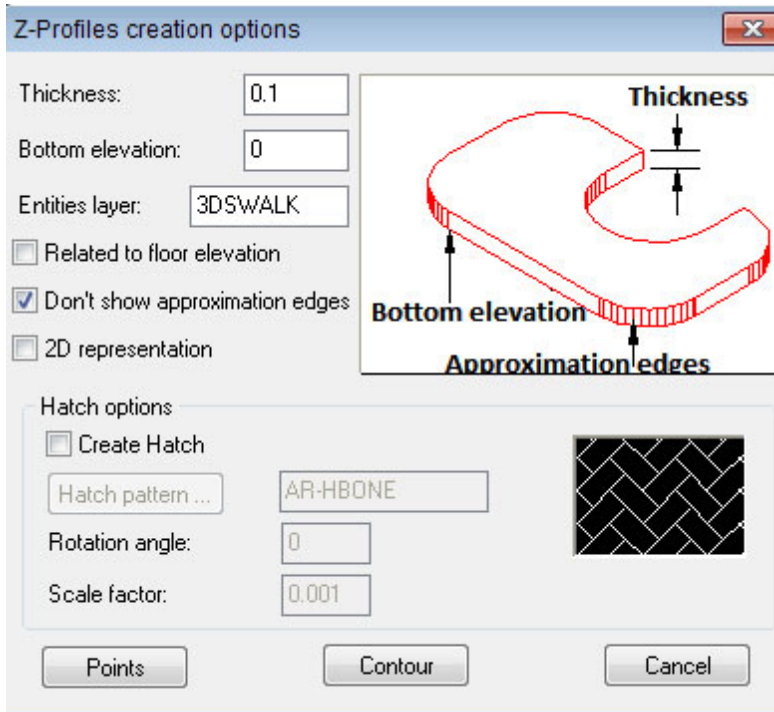
*Block and profile models supplied with AddCAD*

The models illustrated in the following image are provided with the program and can be accessed from the **AddCAD Draw->3D Models->Blocks for railings** menu.



Creating 3D elevation structures

The ZPROF command allows you to generate three-dimensional structures extruding a selected contour. This structure can have as many holes as you wish and can be inclined with respect to the work plane. It could be like the [floor slab command](#) and [roof slab generation](#). But unlike these, its use is more general and therefore allows you to model sidewalks, slides, tracks, parking lots and similar structures. The dialog box allows you to provide the necessary parameters with values.



Thickness

To assign thickness to the element.

Bottom elevation

Elevation of the bottom surface of the element. The possibility of being able to specify the base Z elevation beforehand allows you to establish the elevation of the selected points. The selected points are therefore always cast on the Z plane = *Bottom elevation*.

Related to floor elevation

If the check box is enabled, then the specified base Z elevation is related to the floor elevation of the current floor. If it is not enabled, then the value refers to the UCS World reference

system. If the option is enabled, the object will be considered as a [current floor object](#).

Entities layer

In this box you can change the layer name of all the 3D entities belonging to the object. The entities layer of the 2D representation can be obtained by replacing the 3D prefix with 2D.

2D representation

If you decide to create the 2D representation, the object's contour lines will also be created.

Don't show approximation edges

The curved sections are approximated with flat surfaces. The value of this approximation can be modified with the [SCALEF command](#). By enabling this option, you can avoid viewing the vertical edges of the elements which approximate the curved sections.

Create Hatch

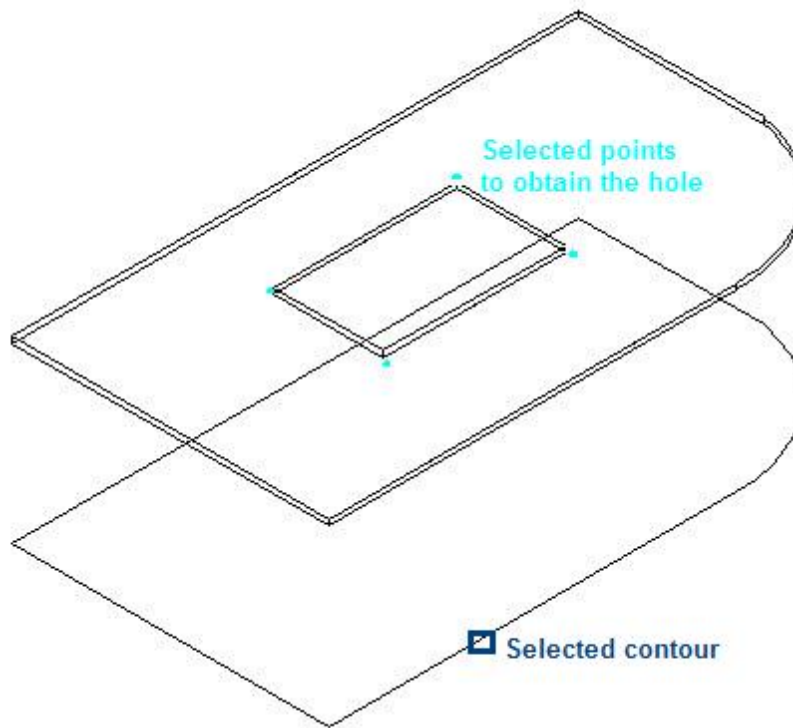
If the *Create hatch* check box is enabled, the command also creates the hatch on the top surface. If you decide to create the hatch, then you can select the model, the rotation angle and the scale factor of the hatch. The hatch can be applied later on using the [HROOF command](#).

The dialog box of the command provides three exit buttons: *Points*, *Contour* and *Cancel*.

With the option *Contour* it is possible to select a closed contour composed of common polylines, lines and arcs or a circle.

The option *Points* should be chosen when you wish to manually and freely indicate the contour with points of the drawing.

An example of this command is shown as in the following example.

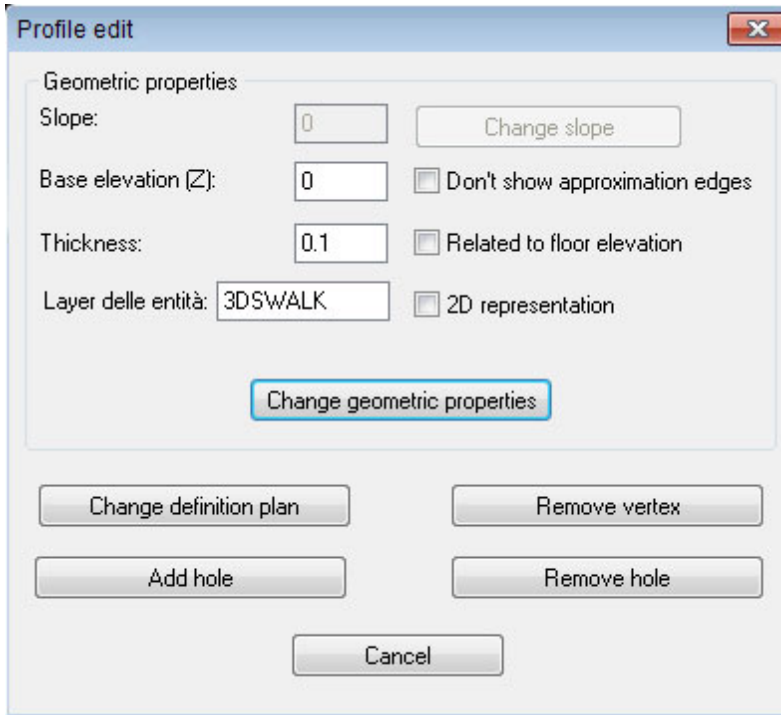


Command: ZPROF
 Select the external closed contour: <contour selected>
 Specify the first vertex of the hole [Contour/Invisible]:
 Specify the vertex number 2 or Enter to finish
 [Invisible/Undo]:
 Specify the vertex number 3 or Enter to finish
 [Invisible/Undo]:
 Specify the vertex number 4 or Enter to finish
 [Invisible/Undo]:
 Specify the vertex number 5 or Enter to finish
 [Invisible/Undo]:Enter
 Specify the first vertex of the hole [Contour/Invisible]:
 Enter

The option *Invisible* allows you to generate the next section with an invisible edge. The option *Undo* cancels the selection of the last point in case of an error.

Modifying 3D elevation structures

The EZPROF command allows you to modify the properties of the elements made with [ZPROF](#). You can give the elements a slope angle, add holes, eliminate vertices and holes.



After having selected the element you wish to modify, the dialog box is displayed with all the current parameter values. The subsequent requests of the command will depend on the type of modification you wish to make.

Change geometric properties

If you choose to modify the properties and not the shape of the element, the command will make no more requests. By properties we mean the visibility of the approximate edges of the curved contours, whether or not there is a 2D representation, relation with the floor elevation, the thickness of the element and the 3D entities layer.

Change slope and base Z Elevation

Having a slope and a base Z elevation are mutually exclusive properties. In the sense that with a constant Z elevation, there is never any slope and with the slope other than zero there is more than one base Z elevation. To provide an element with a slope which does not have one, you must Change the definition plan.

Add or remove hole

For example it is possible to bore a hole in an object by selecting a closed contour or indicating a certain number of hole vertices. You can define a hole also by selecting an object with the relative definition, such as stairs. You can close an opening by selecting a vertex which is part of the opening.

Remove vertex

If you choose to eliminate vertices, the command asks you to select which ones. The vertices can either be on the external contour or on any internal openings. If less than three vertices are remained on an opening, it is completely closed. To add vertices, you may use the [EDITROOFP command](#) (modify and insert vertices).

Change definition plan

You are requested for three points which change the definition plan of the element. Generally this option modifies the slope. The first point is always fixed and does not move, the second can either remain or can be moved together with the third. Since movement also involves the Z coordinate of the point, an element is obtained which changes the definition plan.

Note regarding elements generated with ZPROF and the modification of roof element vertices.

3D elevation elements generated with ZPROF can be modified by using the [EDITROOFP command](#). You can therefore insert new vertices and move existing vertices without changing the general nature and other properties of the component.

Generating polygonal surfaces

The PSURF command allows you to generate 3D polygonal surfaces. AutoCAD also allows you to do something similar using the 3dface command, but this enhanced command allows you to draw surfaces with more than four sides.

The command requests the layer where the entities are to be inserted. The layer can be indicated by selecting an entity. If you press *Enter* the surfaces are inserted on the current layer.

Command: PSURF

Select an object on layer you want use or Enter to use current layer:

Select the first point of the surface:

[Undo]Point number 2:

[Undo]Point number 3:

.....

The command generates polygonal surfaces which rest entirely on a floor. The floor is calculated by using the first three unaligned points indicated. All the other points are cast on the floor that has been obtained, regardless of their coordinates.

Edge visibility

The VISEEDGE command modifies visibility of the edges of the 3dface entities and pface entities. 3dface entities can belong to blocks as is the case for roofs, parapets, eaves and parametric objects. The command simply asks:

Command:VISEEDGE

Select the edge you want make invisible:

Selecting the command switches its status. If the edge was visible before using the command, afterwards it will be invisible. Vice versa if it was invisible it will become visible.

Viewing invisible edges

The invisible edges can be selected by setting the AutoCAD SPLFRAME variable to 1 and regenerating. This way AutoCAD also shows invisible edges which can therefore be selected. After the VISEEDGE command, you must set SPLFRAME to 0 and regenerate. Often roofs, parametric objects and parapets have overlapping visible 3dface edges. In this case to make the edge invisible, you must press VISEEDGE twice.

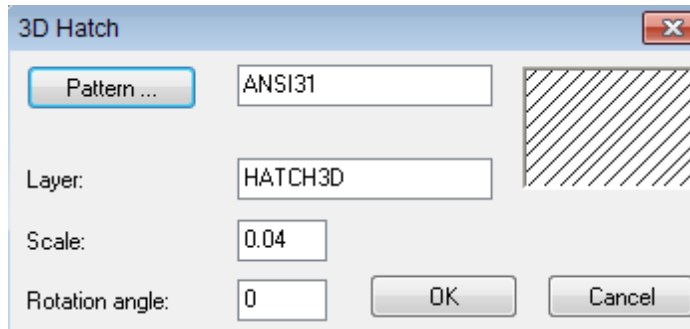
Hatch in 3D space

To hatch 2D elements, such as plan wall segments, AutoCAD has the associative hatch command BHATCH.

The hatches of 3D elements such as [wall faces](#) and [roofs](#) are facilitated and controlled by the AddCAD associative hatch system.

The HATCH3D command allows you to create hatches on inclined planes however they

are arranged and with a number of islands however complex they may be. An example of the application of this command is the hatch of external accessories of a building or of inclined elements. The difference between this command and the BHATCH command of AutoCAD is that HATCH3D allows you to define the hatch surface without modifying the UCS.



You can specify the destination layer of the hatch entities in the dialog box.

Definition of angle and hatch plane

The rotation angle of the hatch is relative to the first two points of the external polygon selected. Whereas the hatch plane is identified by the first three points. The rest of the points are cast on the defined plane by these first three points. Internal islands are

optional. If you press *Enter* to the request of the first point of the first island, the command will immediately switch to the hatch phase.

When inserting the points, the command always links the previous point with an elastic line using the cursor. At any time you can press *Undo* to go back. The *Undo* option can return to the first point if repeated several times.

Command: HATCH3D

Select the external perimeter of the area to hatch:

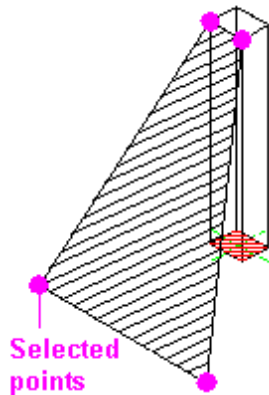
Specify the first point of the hatch boundary:

[Undo]Point number 2: Endpoint of

[Undo]Point number 3:

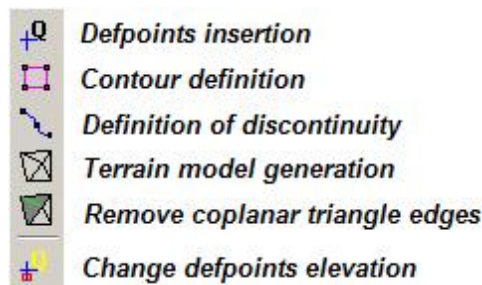
[Undo]Point number 4:

[Undo]Point number 5: Enter



Digital Terrain model

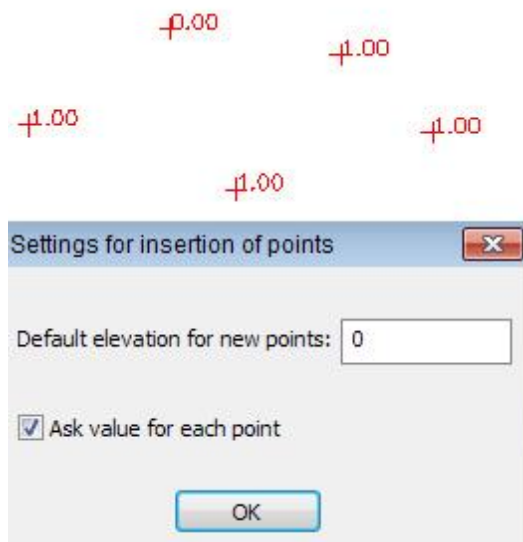
Digital terrain model



The terrain model and external accessories in general are generated by transforming a

set of elevation definition points into a [set of triangular surfaces](#). This transformation is achieved with the Delaunay triangle algorithm which aims at making a network of triangles as equilateral as possible. Creation of the terrain model starts with the [definition of the elevation points](#) consisting of a set of points each of which is assigned with a Z elevation. A [closed external contour](#) must be defined which bounds the entire model area. If necessary, internal contours can be defined inside the points, in which case the triangular surface model is not created. Practically the internal closed contours represent empty spaces. Finally [discontinuity lines](#) can be traced which place constraints on the model generator. These elements actually represent precise orographic characteristics which make it possible to define real paths such as roads, ditches, support walls and so on. Let us now take a look at the various tools available to define the terrain model.

Elevation point



The elevation points are drawn with the DTMPNT command which allows you to insert points in the drawing with a Z elevation value. The command is designed to upload points as fast as possible. The dialog with the user allows you to change the value of the elevation at each point inserted, to insert a sequence of points all at the same elevation without requesting the elevation value each time. Two requests are made:

Elevation <0.00>:

[Settings]Position of elevation defpoint (h=0.00):

The first request is answered by assigning the current elevation value. In response to the second, you can assign the position of the point in the drawing with all the methods allowed by AutoCAD. If you choose the *Settings* option, a dialog box is displayed where you can temporarily disable the

continuous request for the elevation value for each point.

Symbol and elevation value of point

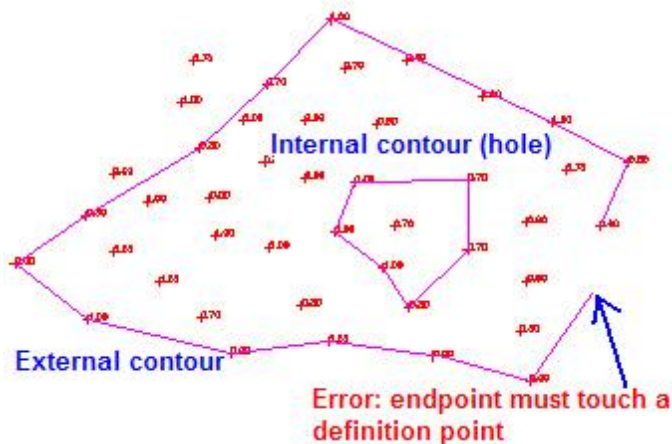
For each point indicated in the drawing, the DTMPNT command inserts an ADDCPDTM name block as shown in the figure. It is actually a small cross with a text indicating the elevation value. The elevation symbol can be customized by modifying this block. Furthermore the cross and text are placed on a layer called ADDCPDTM. Therefore it is possible to disable all the points and to customize their appearance by modifying the features of the layer. The size of the symbol inserted in the drawing depends on the value *Scale factor for symbols* which can be modified with the [scale factor management command](#). Keep in mind that the scale factor management command does not allow you to modify the size of the symbols already inserted. To do this, you can intervene directly on the scale factor of the object properties.

Modifying elevation value of points

The elevation value is a block attribute. The number of decimal places can be modified by intervening in the [Quantities survey system](#) and changing the *Decimal places* value. The

number can be changed by simply using DDATTE, a standard AutoCAD command, or double click to start the Enhanced Attribute Editor, another standard AutoCAD command or use the Edit defpoint elevation toolbar button.

Terrain model contours



Drawing contours helps to set a boundary to the application area of the algorithm which generates the top surface of the terrain model. There must always be a contour, intended as an external contour. Otherwise the terrain model cannot be generated. It is possible to define as many closed contours as you wish, inside the external one, to create empty spaces such as building perimeters. The contours are made of lines which must have endpoints on the topographic points of the elevation

point. The points outside of the external contour and inside the internal contours are disregarded by the terrain modeler. The lines of the contour are normal lines placed on a specific layer called ADDCCDTM.

Command for drawing contours

We recommend using the specific DTMCONT command to draw the contour lines. The DTMCONT command, which can be started directly from the toolbar, automatically activates the *osnap Node* to insert on topographic points and places the lines directly on the right layer. The command makes the following requests:

Command: *DTMCONT*

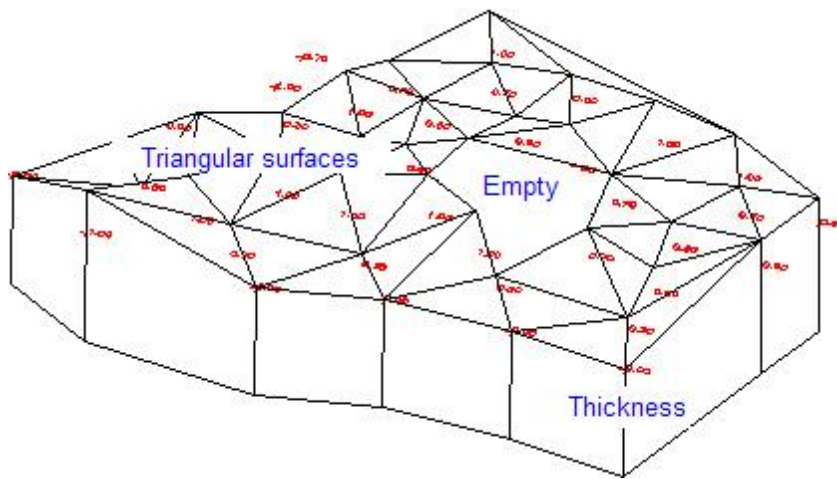
Specify first point:

Specify next point or [Undo], Enter to exit:

....

By means of the *Undo* option, the command allows you to go back one step and to end insertion of the lines by pressing *Enter*.

Calculating triangular model



The DTMTRIANG command generates the digital terrain model. The command stops if there are no contours or if the contour lines and discontinuity lines are not drawn correctly. In this case, the isolated lines are highlighted in the drawing so that these mistakes can be corrected quickly. If the contours are recognized correctly, the command requests the thickness of

the terrain model and then generates the terrain model by applying the Delaunay algorithm thus obtaining a triangulation of the terrain made up of triangles as equilateral as possible.

Command: *_DTMTRIANG*

Terrain thickness <3.000>:

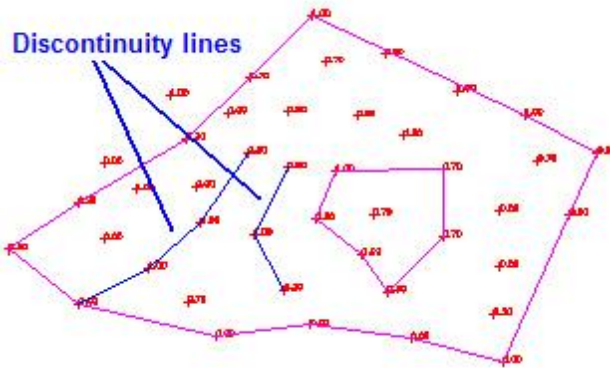
If zero is assigned to terrain thickness, the side thickness surfaces and bottom closing surface will not be generated. Otherwise the top triangular surfaces, the side thickness surfaces and the bottom closing surfaces will be generated. The program places the three groups of surfaces on different layers: 3DDTMODEL for the triangular surfaces, 3DDTMODEL_S for the side delimitation surfaces and 3DDTMODEL_B for the bottom delimitation surfaces.

Eliminating a previous model

The command removes any models generated previously thus automatically updating the model with the new elevation point.

In practice, all the faces of the model belonging to the three layers seen above are canceled from the drawing. If you do not want them to be removed automatically, just change the layer name of the entities generated in the previous model.

Discontinuity lines



Discontinuity lines allow you to introduce constraints which the terrain model generator must take into account. They represent the places of the points where the elevation is known. Discontinuity lines link two points of the elevation point and force the system to have that line coincide with a common side of two adjacent triangles. Discontinuity lines must be on a specific layer called ADDCDDTM and must finish on elevation points. The DTMDISC command, similar to that for inserting

contours, allows you to insert discontinuity lines by placing the lines directly on the correct layer and setting the object snap in automatic *insertion Node* mode.

Command: DTMDISC

Specify first point:

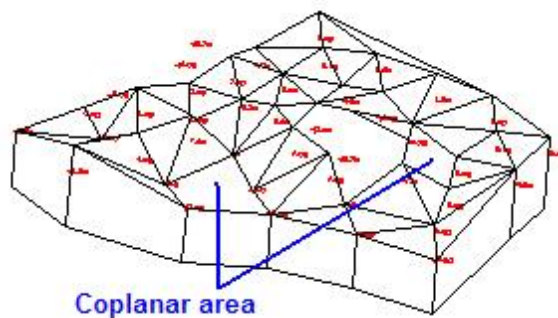
Specify next point or [Undo], Enter to exit:

....

By using of the Undo option, the command allows you to go back one step and to end insertion of the lines by pressing Enter.

Viewing coplanar areas

The DTMPLANE command processes one or more terrain models in the drawing in order to make all the common edges of the two triangles which are coplanar invisible. The command does not make any further requests.



Introduction to frame definition

The AddCAD parametric library has a large number of openings and frames which can be parameterized and inserted in walls.

In most cases, you only need to choose the appropriate object and to assign the parameters so that the correct frame is generated.

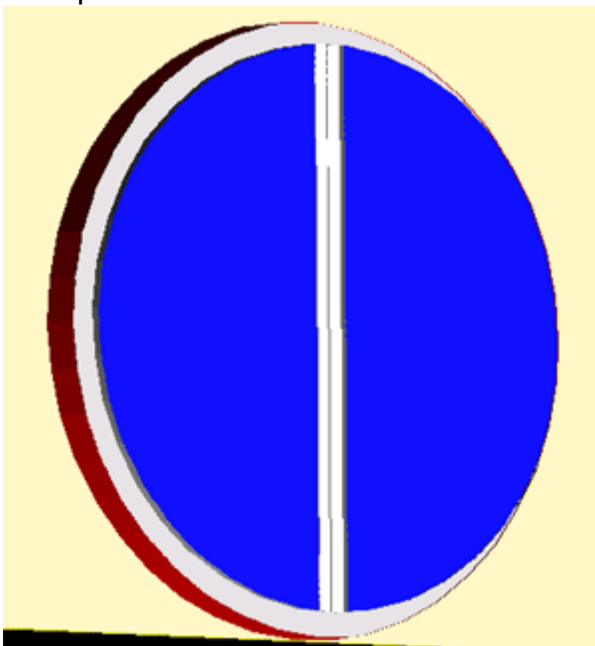
But sometimes we don't find the frame we're looking for in the library. In this case, we must use the frame modeler functions to create one according to particular design requirements. If it has to do with putting a simple hole in the wall, just use the [WHOLE command](#) which, by simply selecting a closed contour, allows you to find what you are looking for. If instead you need an actual frame, you must proceed by designing a new one. We recommend creating the definition of a frame in a new drawing and not the one where the architectural design is being drawn. This will allow us to return to the frame at any time and to quickly change its definition if needed.

The definition of a frame consists of drawing it in the front view using the common lines, arcs, circles and polylines of AutoCAD.

The external perimeter of the hole must be a polyline or a circle.

In this drawing you must assign the individual regions with the thickness properties, the position inside the wall and the layer used for the graphical entities. Fundamentally you define the characteristics of the material of the area. There is a specific FRAMEL command to do this.

Once the layout has been drawn with the definitions of the areas, a definition file must be created to be able to upload it in our architectural drawings like any other parametric object. The command to create the definition file is [FRAMEDEF](#). The easiest way to upload a parametric object is to use the OBJECT command followed by the filename. On the following pages we will describe the definition of labels and the creation of frame definition files, we will use the creation of the circular window shown in the figure as an example.



Holes in walls

To create holes in walls without frames, such as passages of any shape, just draw the closed profile in the drawing and insert it in the wall using the WHOLE command. The profile can be made with a closed polyline, a circle or a set of lines and arcs which describe the closed contour.

The WHOLE command asks you to select the profile and the insertion point of the opening. The Y coordinate of the indicated point determines the Z elevation for inserting the hole. In particular, it represents the insertion floor elevation and the hole is positioned at a height equal to distance in Y between the indicated point and the hole.

The other two requests are similar to those for the insertion of any parametric opening by AddCAD. The viewing direction of the internal face of the hole, from the front, is drawn in profile.

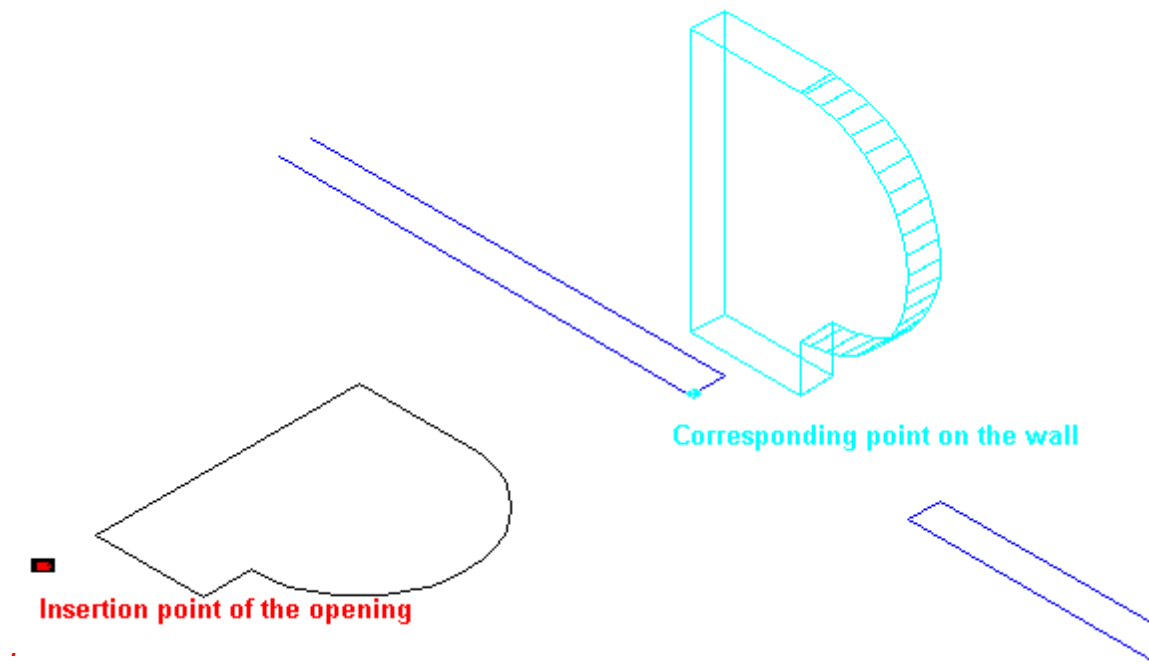
Command: WHOLE

Select the profile:

Specify the insertion point of the opening:

Select the inside line of the wall or Enter to exit or [Parameters/Replace]:

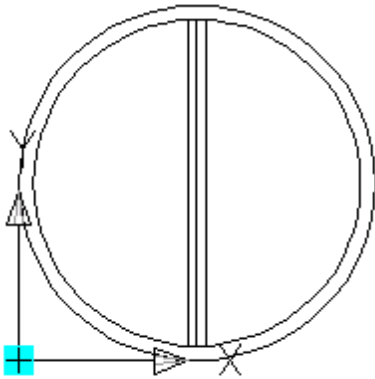
Distance or [wall Face/Reference/Change fixed point/Parameters]:



:

Layout definition and frame labels

In the example shown in the figure we wish to define a circular shaped frame with two opening doors.

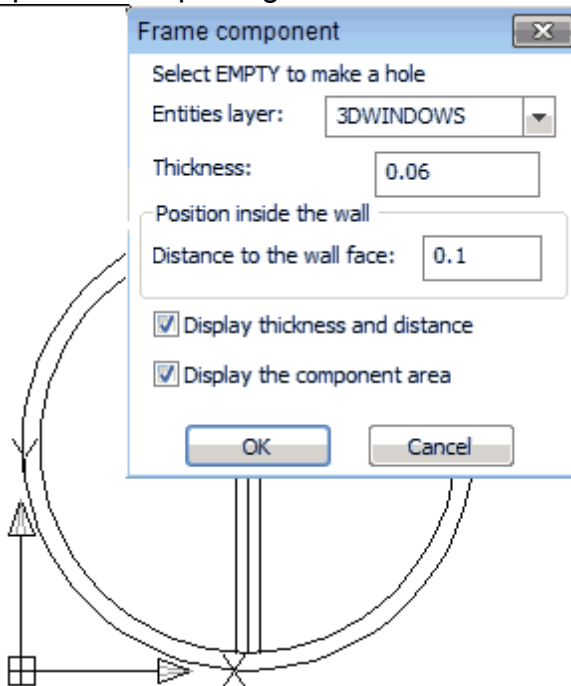


The origin of axis will be the insertion point

You can see that the position of the drawing is important to determine the point of insertion of the frame in the wall. If for example we want the bottom quadrant to be positioned in the wall at zero elevation, it must be at Y equal to zero. If we have a window with a windowsill at 90 cm, we must position the line in the layout representing the windowsill at $y = 90$ cm.

Note concerning unit of measurement to be used

As usual with AddCAD, the unit of measurement used depends on the settings in *Parametric and stairs layer tab* of AddCAD options. If we are working in centimeters, it means that if I want a frame 4 meters wide, I must draw 400 units. If however I am working in meters, I have to draw 4 units. When we upload the frame in a drawing, it will always be uploaded respecting the unit of measurement of the drawing.



Inserting frame labels

Frame labels help to match a layer, a thickness or a position inside the wall to an area of the frame model. Let's look at the layout in the figure. For example, we want the circular border to be 6 cm thick and for it to be generated on the 3DWINDOWS layer. To assign this feature to the area, we can use the FRAMEL command.

FRAMELE opens a dialog box which allows us to choose this information. The fields are:

Entities layer

Entities layer of 3D surfaces which will be generated during insertion of the frame. You can choose a layer from the list or write a new one.

Thickness

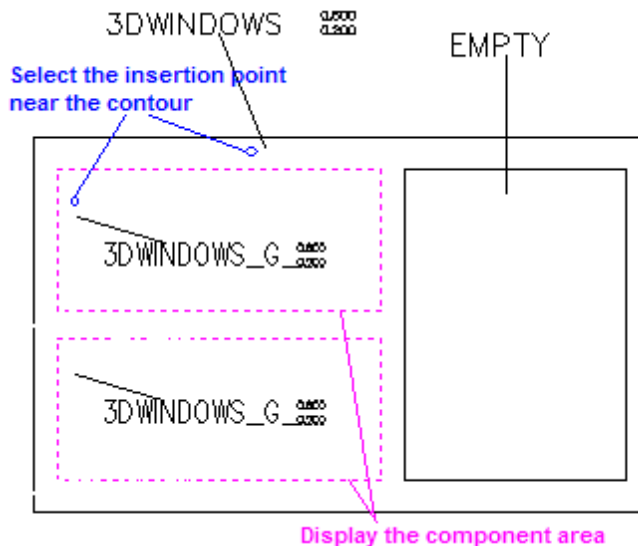
Thickness of the area.

Distance to the wall face

The distance to the wall face. Whether it is the external or internal one is determined when writing the file because it is valid for all the defined areas.

Display thickness and distance

The label is just the layer name linked to a leader in the area it defines. It may or may not indicate the distance thickness.



Display the component area

If this option is enabled, when the point is indicated, the program highlights the recognized area of the region.

Empty holes in frames

Only if there is a label inside can an area be recognized, in order to make an empty hole without generating entities, a label must be inserted with the special layer name *Empty*.

Points required while inserting the label

Once you have finished by pressing OK, the command requests two points. The first must be inside the region you are defining and the

second is the position of the label. Keep in mind that the label leader must be inserted as close to the area contour as possible. The command is repetitious, it allows you to insert several labels of the same type while requesting the point inside the region and the point of insertion of the label each time. Pressing *Enter* ends the insertion cycle.

Modifying labels already inserted in drawing

Once the labels have been inserted, they can be modified using the EFRAME command. The command, after having selected the label to be modified, opens the dialog box and allows you to change the data. Once you have finished by pressing OK, you are only asked for the new position of the label if you wish to change it, otherwise you can respond by pressing *Enter*. The labels can be moved or copied by moving the endpoint of the leader to the desired region.

Command: EFRAME

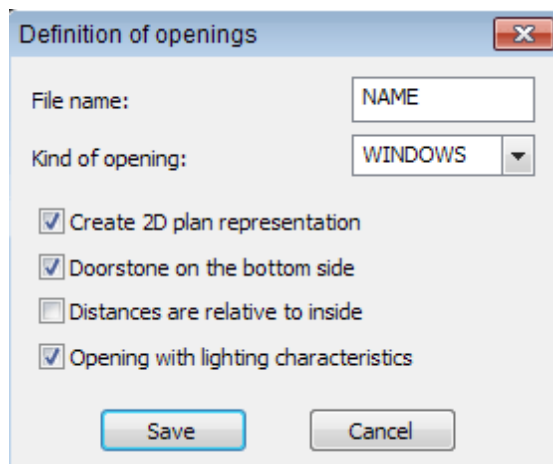
Selezionare etichetta infissi da modificare:

<finestra di dialogo per modifica valori>

Nuovo punto inserimento testo o Enter per confermare l'attuale posizione:

Frame definition

The FRAMEDEF command creates the definition file of the frame object.



First of all you must specify some options regarding the object in the dialog box.

File name

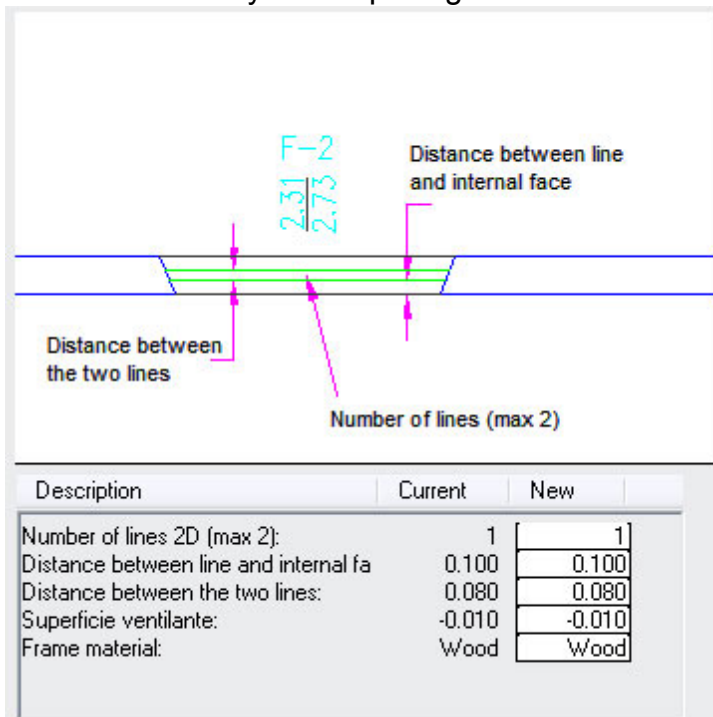
The filename of the object without any extension. You must write this name to answer the OBJECT command when you wish to insert the object in the drawing. Remember that the file is saved in the *USERLIB* folder which is in the main folder of the program. We recommend saving the frame drawing in this folder as well.

Important: Make a backup copy of this folder.

Kind of opening

The value selected is used to create the block layer name. In practice, the block reference to

this object is placed on this layer after having transformed the name by putting ADDC at the start and adding the floor elevation at the end. Furthermore, this value is used to decide whether the frame belongs to the window, door or generic opening family in order to insert it correctly in the opening tab.



Create 2D plan representation

If this option is enabled, all of the entities representing the 2D plan frame are inserted in the frame definition file.

Besides the frame dimensions and codes, a simple 2D plan representation which can be parameterized is included.

The parameterization dialog box illustrates the 2D plan model which is included in the

frame object definition should the option Create 2D plan representation be enabled.

Doorstone on the bottom side

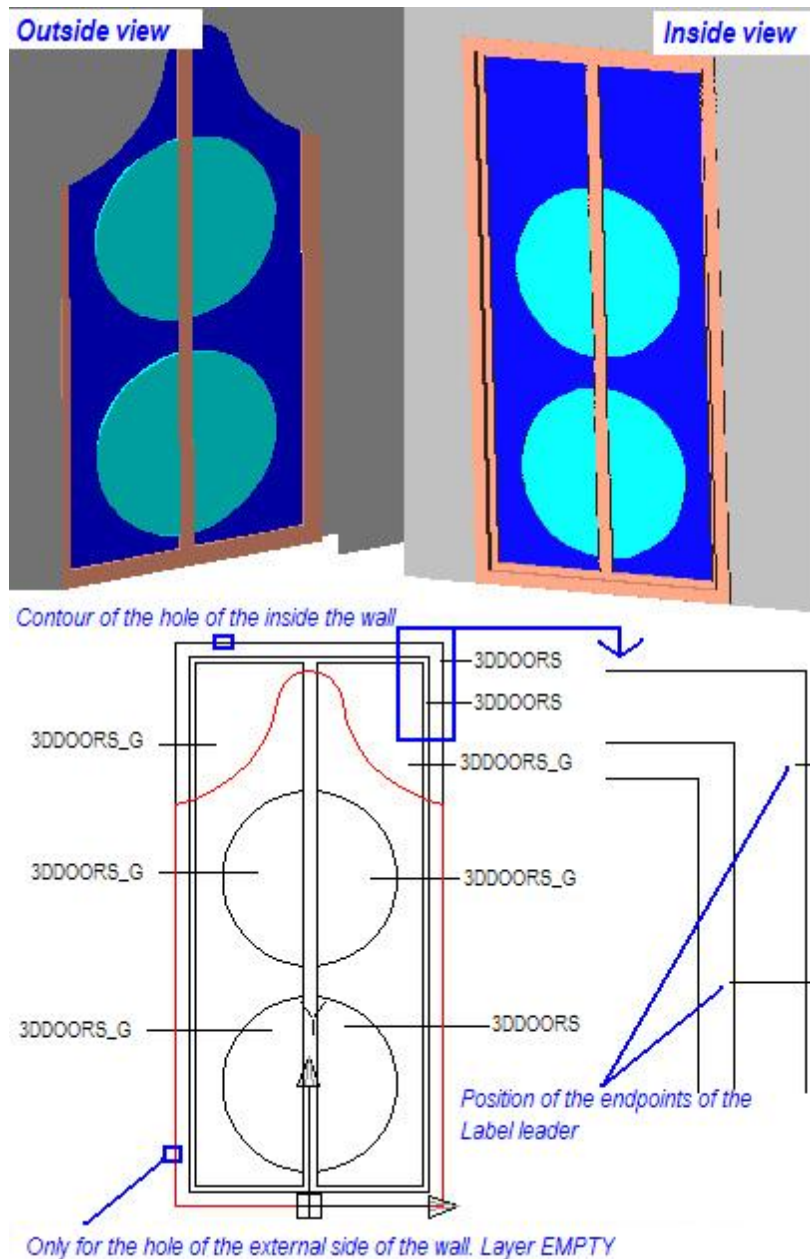
If the opening ends at the floor, disable this option because the discontinuity lines between the wall and floor will not be drawn.

Distances are relative to inside

While an opening is being inserted, AddCAD asks to select the internal wall face. During definition of the labels, distances from the wall face have been indicated. Here we can choose whether the distances refer to the internal or external wall face.

Opening with lighting characteristics

This property can be attributed to the frames. This option will be activated for windows, but not for doors. The entire area of the hole is considered lighting if this option is activated.



Having exited the dialog box, the following request is made.

Select a circle or polyline of the hole inside the wall or Enter if without holes:
Select a circle or polyline of the outside hole or Enter if the same as inside hole:

It is therefore possible to define none, one or two hole polylines, one for the internal face and one for the external face. Remember that if there is only one hole for both faces, the frame model will be completed with the closure on the side border of the wall. Whereas if there are two polylines, you must define the required closing elements.

Polylines for only hole

Provided with a model label, AddCAD tries to determine its region of definition, taking the entity closest to the label insertion point as a component of the contour. In general this function can work but in some cases it creates problems. If we have a polyline with which we would like to simply indicate a hole in the wall and

therefore is not an element of the frame structure, some confusion could arise. To avoid this problem, we can place the polyline of only the hole in the wall on a specific name layer, namely called *Onlyhole*. No label will be able to interpret it as its region of definition.

Introduction

To draw three-dimensional roofs and coverings, AddCAD has a series of commands and a series of objects. A hip roof is generated fully automatically by the [HIPROOF command](#). A gable roof, with or without side slabs, is made automatically with the [GROOF command](#). Roofs on curved walls can be implemented automatically both by inserting parameterizable objects which create the roof by inserting a cone sector, and with the HIPROOF command by selecting the external roof contour, which can include curved sections. It is possible to insert [parametric objects on roofs](#), in particular openings, gables, skylights, etc. The [R3D command](#) is definitely the most flexible allowing you to draw complex roofs freely and with extreme ease. The [FSLAB command](#) draws horizontal partitions.

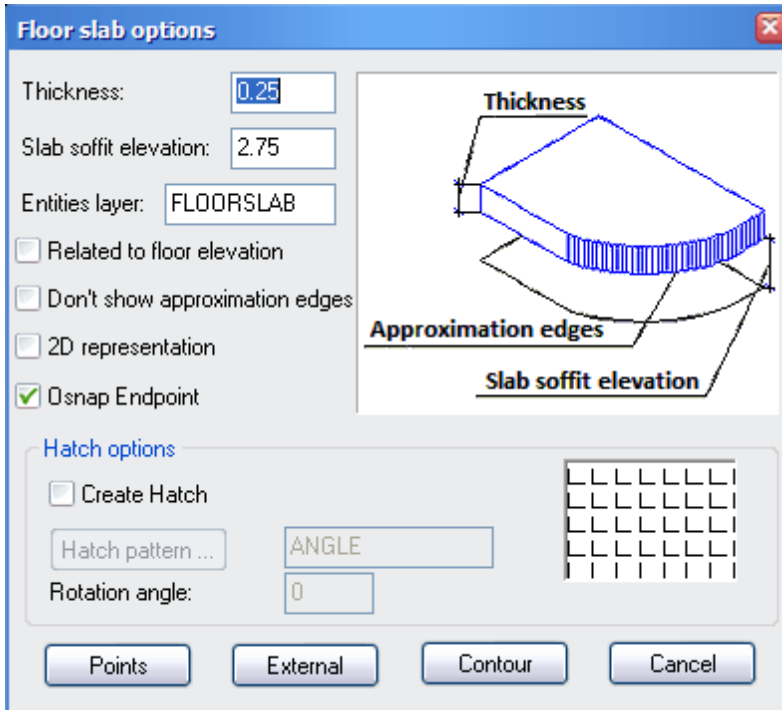
The [EDITSLABS command](#) is a slab editor, allowing you not only to change a series of roof and floor slab properties, but their geometry as well. Other commands and functions are available to modify and build complex roofs. For example often you need to [find points on a roof slab](#) which represent the exact position at a certain elevation. There is a command which does so by tracing under-roof 'isohypse' lines. There is still another command which allows you to find slab intersection points with other objects. These points create lines which can be used for example to join slabs of different roof systems. It is possible to [add and move slab vertices](#) quickly without needing to pass through the roof editing dialog box.

Many roof drawing commands have the option of generating hatches simultaneously as the roof slabs are drawn. Often it is not advisable to create the hatch right away, as hatch entities are very cpu intensive. Alternatively the hatch can be generated later on, again automatically using the [HROOF command](#). The three-dimensional roof model is also a surface model based on 3dfaces. If necessary, a 2D roof representation can also be generated.

Curved roofs such as [barrel vaults](#) and [cross vaults](#) can be made and it is possible to apply [girders to the roof slabs](#) to draw not only concrete roofs but even in wood and pergolas.

Floor slabs

The characteristics and options of the FSLAB command make it suitable to build floor slabs. The building parameters can be set inside a dialog box.

**Thickness**

The edit box allows you to assign the thickness to the floor slab.

Slab soffit elevation

From a graphical viewpoint, it represents the elevation of the bottom slab surface. The possibility of being able to specify the soffit elevation beforehand allows you to establish the elevation of the selected points. Even if the object snap is enabled, the elevation is not affected by the point selection. The selected points are therefore always cast on the Z plane = *Slab soffit elevation*.

Related to floor elevation

If the check box is enabled, then the specified slab soffit elevation is related to the floor elevation of the current floor. If it is not enabled, then the value refers to the overall UCS World reference system. If the option is enabled, the floor slab will be considered as a [current floor object](#).

Entities layer and 2D representation

In this edit box, the common layer root generated by the command can be written. AddCAD generates different layers when it creates the floor slabs. If for example FLOORSLAB is written, three entities layers are created: 3DFLOORSLAB_T, 3DFLOORSLAB_B, 3DFLOORSLAB_S respectively for the top, bottom and side surfaces. If *Related to floor elevation* is enabled, the name of the layer where AddCAD will place the floor slab block will be joined with the floor logic. Therefore the floor slab will be visible if the floor which it belongs to is visible. If you decide not to relate the floor slab to the floor elevation, then the block layer will simply be the common root inserted in the edit box. If you decide to create the *2D Representation of the floor slab*, its contour lines will also be created. These lines will be placed on the 2DFLOORSLAB layer.

Don't show approximation edges

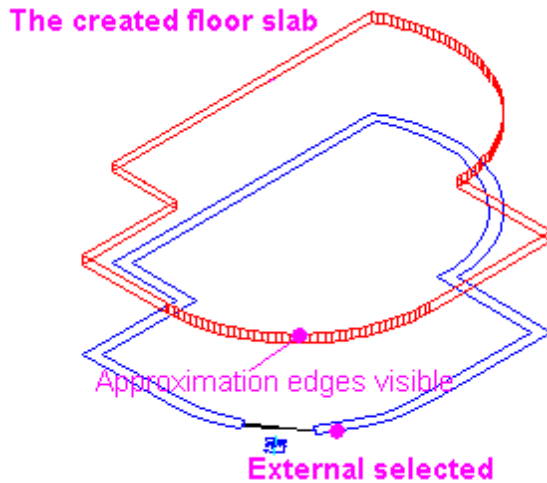
The curved sections of the floor slabs are approximated with flat surfaces. The value of this approximation can be set with the [SCALEF command](#). By enabling this option, you can avoid viewing the vertical edges of the elements which approximate the curved sections of the floor slab.

Osnap Endpoint

If you exit by pressing *Points*, the command requests the vertices of the floor slab. Sometimes it is convenient to select these points as *endpoint* of an edge or of the line.

Create Hatch

If the Create hatch control button is enabled, the command also creates the hatch on the top surface of the floor slab. If you decide to create the hatch, then you can select the model and the rotation angle of the hatch. The hatch scale is controlled by the [SCALEF command](#).



The dialog box of the FSLAB command provides four exit buttons: *Points*, *External*, *Contour* and *Cancel*.

The *External* button allows you to select a line or an arc of the external perimeter of the plan distribution. The command automatically calculates the external vertices of the floor slab and then immediately requests if there are holes on the slab.

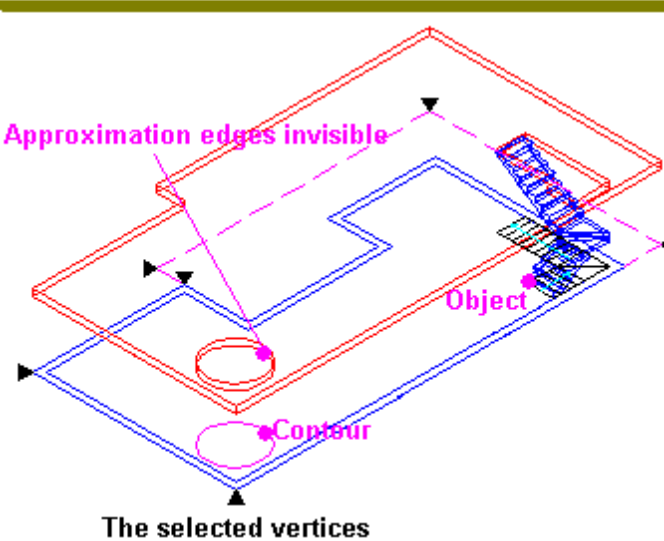
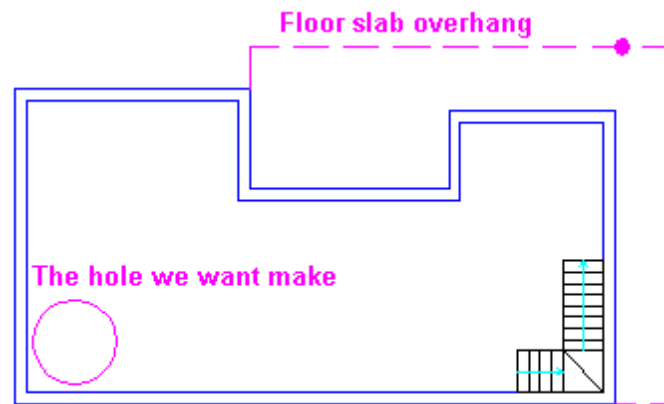
The *Contour* option works in a similar way to the *External* option. The difference is that it is possible to select a closed contour composed of common polylines, lines and arcs or a circle. These options have a great advantage when the slab precisely overlaps the external perimeter of the construction or when, in the

second case, there are polylines or lines and arcs which describe the 2D geometry of the floor slab. When on the other hand the floor slab covers a different area, then the perimeter should be indicated by hand and the *Points* option chosen.

Let's suppose we have the situation in the second figure. In this case, the polygon points of the floor slab need to be indicated.

Once the external points have been indicated, the command requests whether holes need to be made in the floor slab. The roof elements can be created either by selecting contours designed in the drawing, by manually indicating points or by selecting stairs either [generated with the modeler](#) or [parametric stairs](#) or other roof slab openings. Let us suppose that we have to make the indicated holes in the floor slab in the second example. It is a circular hole where the stair passes between two floors.

Having completed the dialog box by pressing *Points*, the contours can be drawn with common polylines, circles or with lines and arcs



Command: FSLAB

Specify the first vertex of the slab or [Invisible]:

Specify the vertex number 2

[Invisible/Undo]:

.....and so on until point 6

Specify the vertex number 6

[Invisible/Undo]:

Specify the vertex number 7

[Invisible/Undo]:Enter

Specify the first vertex of the hole

[Contour/Invisible/Object]:

Object

*Select a stair or a parametric object that defines a hole for the slab:*stair selection

Remember the stairs generally have a parameter called *Headroom height* with which AddCAD is able to calculate the size of the opening on the slab.

Specify the first vertex of the hole

[Contour/Invisible/Object]:Contour

Specify the closed contour of the slab hole: circle selection

Specify the first vertex of the hole

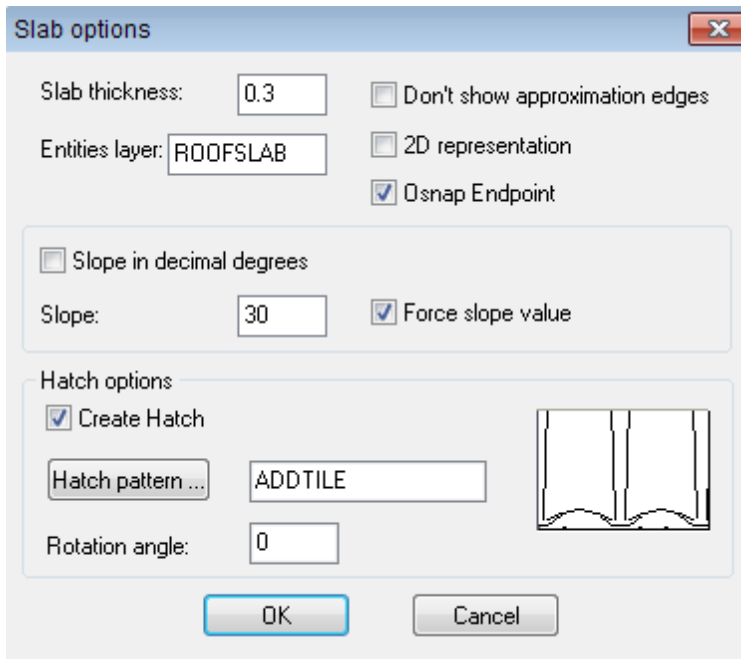
[Contour/Invisible/Object]:Enter

Option Invisible

The FSLAB command also has the Invisible option available. If when requested a point you respond with a letter "I", the top and bottom edges which attach this point to the next one will be invisible. This option can solve the problem for example of joining slabs with different thicknesses.

Single roof slabs

One of the most flexible commands for the generation of roof slabs is R3D. It has a [large range of applications](#). Without a doubt, the most important one is the possibility of quickly constructing extremely complex roofs and coverings. There are many different strategies for drawing roofs, which can lead to the same result. Much depends on the initial project data. This command generates roofs while describing the roof slabs individually. To begin with you might find the system to be inefficient and slow. But as you learn the flexibility of this command, you will understand how powerful it is and that there are no roof slabs which you are not capable of drawing. Let's start by examining the options and requests of the command.



Slab thickness

The edit box allows you to change the thickness of the roof slabs.

Entities layer

It contains the common layer root generated by the command.

Starting from this common part, several different layers are created. If for example we put the name ROOFSLAB in Entities layers, AddCAD will generate three different name layers; namely 3DROOFSLAB_T, 3DROOFSLAB_B, 3DROOFSLAB_S, where the *Top* floor surfaces, *Bottom* floor surfaces and *Side* surfaces are placed respectively. If the 2D representation is generated as

well, the 2DROOFSLAB layer will be created. Furthermore the roof slab block is inserted on the ROOFSLAB layer. This allows you to match the 3dface surface layers with different colors and materials.

2D representation

The check box is used to also create the 2D representation lines of the floor slab contour.

Osnap Endpoint

When describing the geometries of the floor slabs, we often refer to entity points present in the drawing. For this reason it can be convenient to use *Osnap Endpoint*.

Force slope value

Force slope value allows you to draw a floor slab with a known slope. There are situations which utilise this possibility. The slope must be indicated in the edit box to the side. The slope can be inserted in degrees or percentage. The relative check box must be enabled if you wish to insert the slope in degrees. If *Force slope value* is not selected, the roof slab plane can be provided by indicating the three points in space.

Create Hatch

If the *Create hatch* check box is enabled, R3D also creates the hatch on the roof slab. If you decide to create the hatch, then you can select the model and indicate the rotation angle of the hatch. The hatch scale is set by the [SCALEF command](#).

When the window has been closed, the command will continue according to how *Force slope value* was set. The difference lies in how the roof slab plane is selected.

Independent from Force slope value setting

Specify the first vertex of the roof slab or [Invisible]:

Specify the vertex number 2 [Invisible/Undo]:

.....
The command requests to select the vertices of the roof slab contour. There is no limit to the number of vertices, but at least three must be selected. By pressing *Undo* you can cancel the last point entered and with *Invisible* you can tell AddCAD that the side that goes from the point being indicated to the next one must have invisible edges. Pressing

Enter terminates the vertex selection.

Specify the first vertex of the hole [Contour/Invisible/Object]:

The command asks you to indicate the internal openings. It can be done by selecting specific parametric objects such as stairs or roof slab openings, or by selecting closed contours made of the closed polylines or of lines and arcs or a circle. Finally it is possible to indicate contour points exactly like the external polygon. Pressing *Enter* ends the introduction of internal contours.

Force slope value not enabled

First point of the definition plan or Enter to use the first three points:

Second point of the definition plan:

Third point of the definition plan:

The R3D generates flat and inclined roof slabs. This means that all points, either of the external polygon or hole contours, are cast on a floor. A fundamental geometric property of floors is that only three unaligned points are needed to identify it. The floor is defined by the three points selected in the last phase. If the floor is already defined by the first three points of the external polygon, then you can skip this last phase by pressing *Enter* when requested the first point of the definition plan.

Force slope value enabled

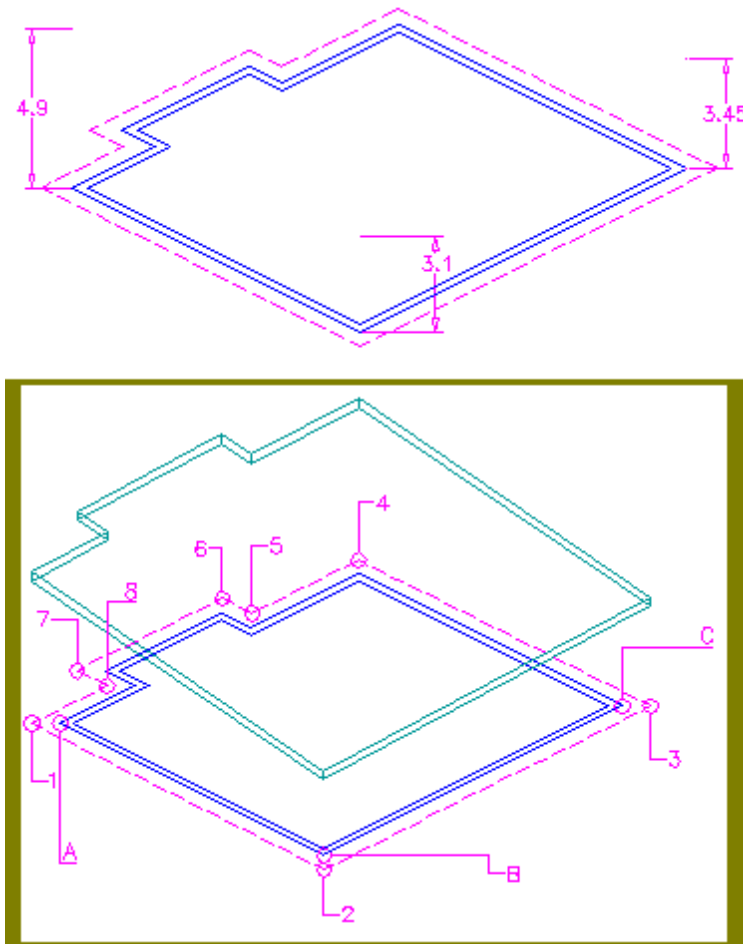
First point of the gutter side, Enter to use the first two points:

Second point of the gutter side:

In this case, you only need to give the two endpoints of the lower roof slab side and the roof slab plane is calculated thanks to the known slope. If you began to define the contour from these two points, you can simply respond by pressing *Enter*.

Normally the casting on the floor is oblique, with the z axis as the casting direction. In fact only the Z coordinate of the points is changed, so that the original X and Y coordinates remain, but resting on the floor.

Examples of roof generation



Example with three points at known elevation

Let's suppose that we have a perimeter in masonry like the one indicated by the blue lines in the drawing. And let's also suppose that we wish to draw just one overhanging roof slab as indicated by the hatched lines, and that the measurements indicated by the dimensions at the angles of the wall are known due to a survey carried out on-site. At this point we can directly launch the command and write the following points.

Command: *R3D*

Specify the first vertex of the roof slab or [Invisible]: Point 1

Specify the vertex number 2

[Invisible/Undo]: Point 2

.... and so on until...

Specify the vertex number 8

[Invisible/Undo]: Point 8

Specify the vertex number 9

[Invisible/Undo]: Enter

Specify the first vertex of the hole [Contour/Invisible/Object] or

Enter to exit: Enter

First point of the definition plan or Enter to use the first three points: .xy

Select XY of: Endpoint A

Still need Z of: 4.9

Second point of the definition plan: .xy

Select XY of: Endpoint B

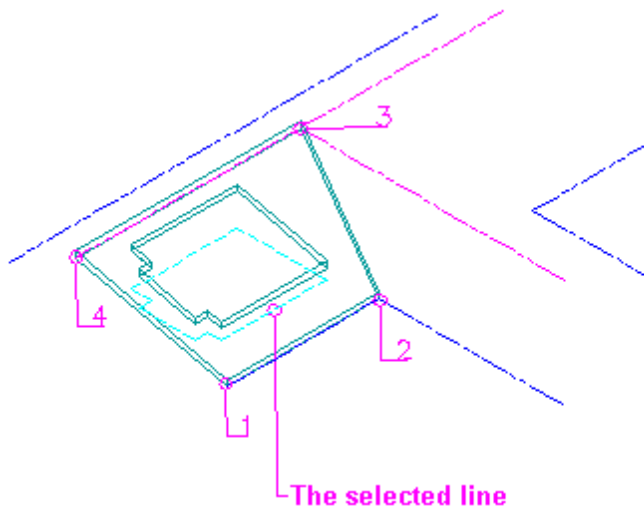
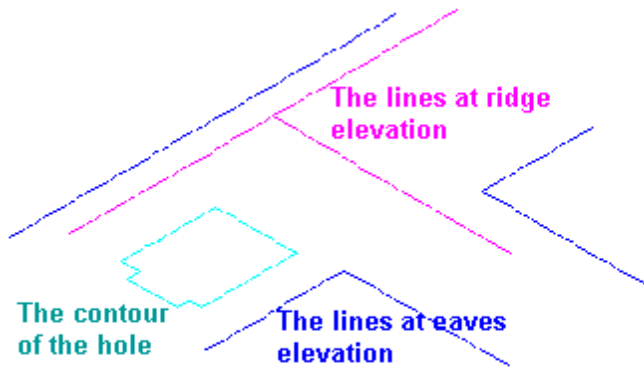
Still need Z of: 3.1

Third point of the definition plan: .xy

Select XY of: Endpoint C

Still need Z of: 3.45

It is often convenient to use the coordinate filter together with the *object snap* to indicate the points in two phases: the first identifies the xy coordinates and in the second we write the Z value.



Example with construction lines

Some helping lines can be convenient for constructing roofs. They can be ridge lines, gutter lines or hole lines. They can be canceled when the roof drawing is finished.

Command: R3D

Specify the first vertex of the roof slab or [Invisible]:Point 1

Specify the vertex number 2

[Invisible/Undo]:Point 2

Specify the vertex number 3

[Invisible/Undo]:Point 3

Specify the vertex number 4

[Invisible/Undo]:Point 4

Specify the vertex number 5

[Invisible/Undo]:Enter

Specify the first vertex of the hole

[Contour/Invisible/Object] or Enter to exit:Contour

Select the closed contour of the roof slab hole: Select line

Specify the first vertex of the hole

[Contour/Invisible/Object] or Enter to exit: Enter

First point of the definition plan or Enter to use the first three points:

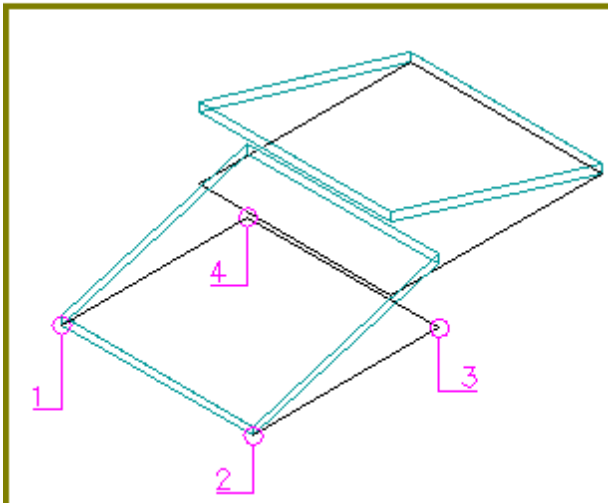
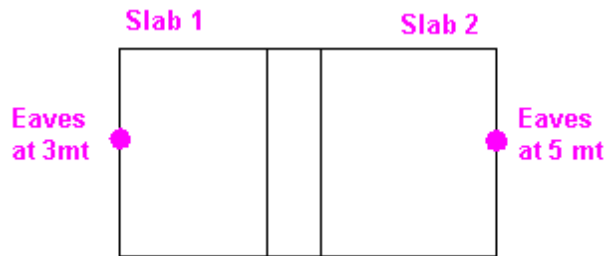
.Enter

To construct the roof, first of all the lines should be brought to the project elevation. The ASSIGNZ command of AddCAD or one of the many AutoCAD commands can be used to this purpose.

As seen in this case, points 1, 2, 3 selected with the Osnap Endpoint already determine the plane of the roof slab. Therefore it is possible to respond by pressing *Enter* when requested the casting floor points.

Example with forced roof slab slope value

In this example, the data of the project which is known is the slope of the roof slabs, which we assume to be 30%. Furthermore let's suppose to know the gutter elevation as well. The top view of the roof slab encumbrance highlights a partial overlapping of the second slab over the first.



In this case as well the lines of roof slab 1 should be brought to 3 meters and those of slab 2 to 5 meters. We can enable the *Force slope* value box in the dialog box of the R3D command and write 30 in the *Slope* edit box. The command requests are slightly different at the end of indication of the points.

Command: R3D

*Specify the first vertex of the roof slab or [Invisible]:*Point 1

*Specify the vertex number 2 [Invisible/Undo]:*Point 2

*Specify the vertex number 3 [Invisible/Undo]:*Point 3

*Specify the vertex number 4 [Invisible/Undo]:*Point 4

*Specify the vertex number 5 [Invisible/Undo]:*Enter

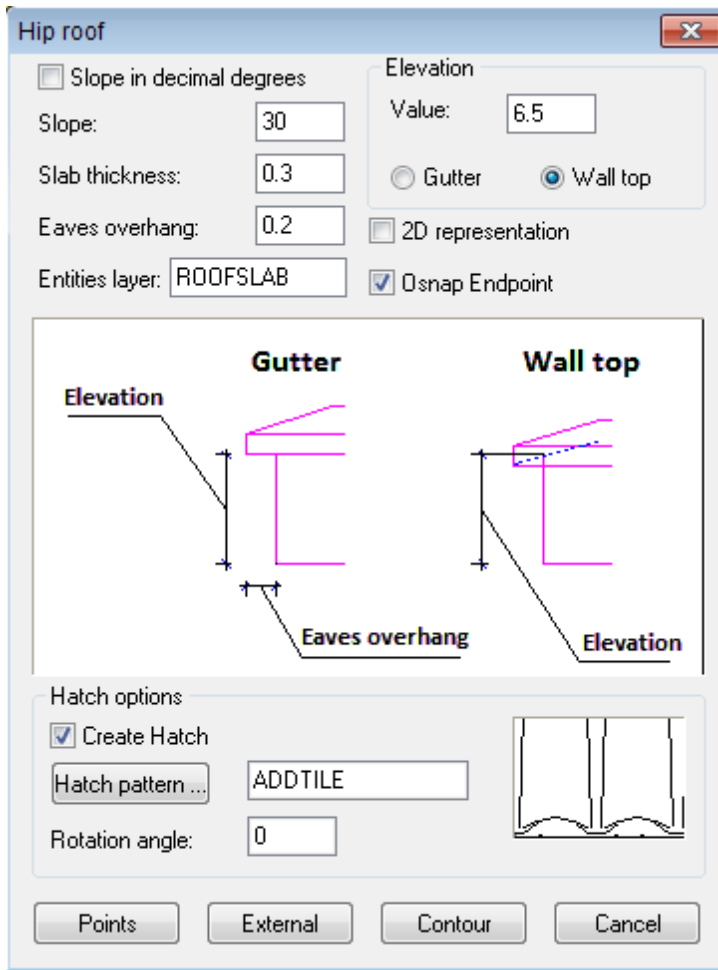
*Specify the first vertex of the hole [Contour/Invisible/Object] or Enter to exit:*Enter

First point of the gutter side, Enter to use the first two points: Enter

Having selected the first two points with the object snap, the gutter side, together with the forced slope, fully determine the position of the roof slab. Just repeat the same operation with the second roof slab. The gutter side must be at a constant elevation. The elevation of the first point is intended as the gutter elevation.

Hip roofs

The HIPROOF command allows you to draw hip roofs. These roofs are defined by the following properties: each gutter segment must identify a roof slab and all the slabs have the same slope. The command is fully automatic, it calculates the contour of the roof slabs automatically and draws them without user input. The dialog box proposes a series of construction options.



Slab thickness

The edit box allows you to change the thickness of the roof slabs.

Slope

The slope of all the roof slabs which HIPROOF will draw must be indicated. The slope can be inserted in degrees or percentage. The relative control box must be enabled if you wish to insert the slope in degrees.

Entities layer

This layer contains the common layer root generated by the command. Several layers are generated and are used later for rendering. If for example we put the name ROOFSLAB in *Entities layer*, AddCAD will generate three different name layers; namely 3DROOFSLAB_T, 3DROOFSLAB_B, 3DROOFSLAB_S, where the Top floor surfaces, Bottom floor surfaces and Side surfaces are placed respectively. If the 2D representation is generated as well, the 2DROOFSLAB layer will be

created. Furthermore all the blocks of the individual roof slabs are inserted on the ROOFSLAB layer. This allows you to match the 3dface surface layers with different colors and materials.

Osnap Endpoint

The Osnap *Endpoint* mode is enabled while selecting the points.

2D representation

The check box is used to optionally create the 2D representation lines of the individual floor slab contour.

Eaves overhang and roof elevation

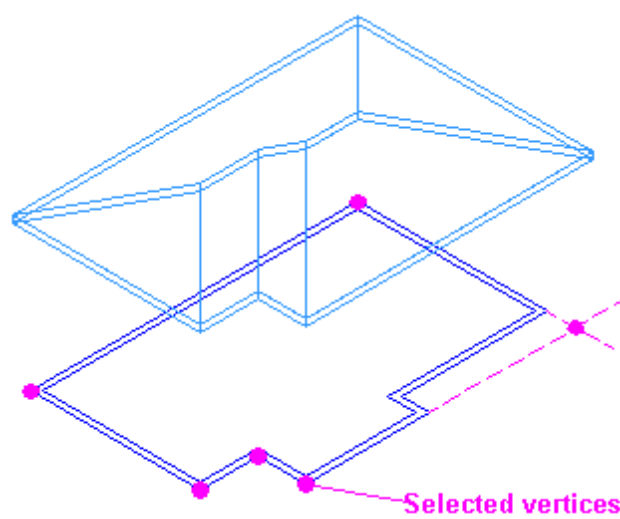
An eaves with specified dimensions overhangs outside of the indicated points and of the selected contour. It is also possible to insert the gutter elevation at the two characteristic roof points. However it must be kept in mind that the value indicated does not consider a floor elevation, but the absolute value of the overall UCS World elevation.

Create Hatch

It is possible to enable *Create hatch*. In this case you can select the model and modify the hatch rotation angle. Its value varies according to the model. The hatch scale is controlled by the [SCALEF command](#).

Once you have accepted the options offered by the command, you must specify the roof plan geometry. There are three different ways of indicating the plan geometry. You can choose between *Points*, *External* or *Contour*.

The example in the figure illustrates the procedure with the point selection.

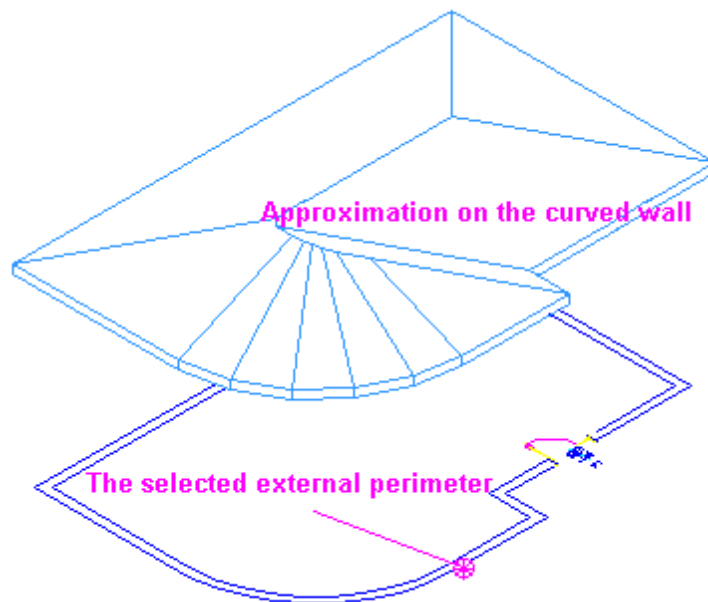


*Specify the first vertex of the hip roof:
Specify the vertex number 2 [Undo]:
Specify the vertex number 3 [Undo]:
Specify the vertex number 4 [Undo]:
Specify the vertex number 5 [Undo]:
Specify the vertex number 6 [Undo]:
Specify the vertex number 7 [Undo]: Enter*

With the External method, you are asked to select the external perimeter of a building. All the corners found become definition vertices.

With the *Contour* method you can select a closed sequence of lines and arcs or a closed polyline. If curved sections are found with the External and Contour

method, then the program approximates the curved trend with four or five flat roof slabs. The program takes a while to search for the roof solution. The time it takes depends exponentially on the number of vertices. The program accepts up to 23 vertices (and therefore a maximum of 23 roof slabs). We have noticed that the calculation is immediate up to 15/16 vertices. You need to wait a few minutes for 19/20 vertices.



Gable roofs

The GROOF command allows you to draw gable roofs automatically. This type of roof can have two or four slabs. The dialog box and modifiable parameters to create these types of roof are described in the following dialog box. The roof is rectangular.

Gable roof

☐ Slope in decimal degrees
Slope: 30
Slab thickness:
Eaves overhang: 0.2
Front overhang: 0.3
Entities layer: ROOFSLAB

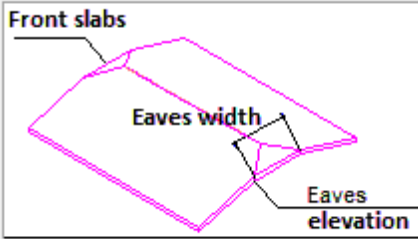

Front slabs
☐ Create front slabs
☒ Force eaves width
Eaves width: 3
Eaves elevation: 6.8
Front slabs slope: 40

Elevation
Valeur: 6.5
☒ Gutter ☐ Wall top

☐ 2D representation
☒ Osnap Endpoint

Hatch options
☐ Create Hatch
Hatch pattern: ADDTILE
Rotation angle: 0

OK Cancel

Slope

The slope of the two main slabs of the gable roof must be indicated. The slope can be inserted in degrees or percentage. The relative control box must be enabled if you wish to insert the slope in degrees.

Slab thickness

The edit box allows you to change the thickness of the roof slabs.

Entities layer

It contains the common layer root generated by the command. Starting from this common part, several different layers are created. If for example we put the name ROOFSLAB in *Entities layers*, AddCAD will generate three different name layers; namely 3DROOFSLAB_T, 3DROOFSLAB_B, 3DROOFSLAB_S, where the Top floor surfaces, Bottom floor

surfaces and Side surfaces are placed respectively. If the 2D representation is generated as well, the 2DROOFSLAB layer will be created. Furthermore all the blocks of the individual roof slabs are inserted on the ROOFSLAB layer. This allows you to match the 3dface surface layers with different colors and materials.

2D representation

The check box is used to optionally create the 2D representation lines of the individual floor slab contour.

Osnap Endpoint

The Osnap *Endpoint* mode is enabled while selecting the points.

Front overhang and roof elevation

An eaves with specified dimensions overhangs outside of the indicated points. It is also possible to insert the gutter elevation at the two characteristic roof points.

Create Hatch

It is possible to enable *Create hatch*. In this case you can select the model and modify the hatch *rotation angle*. Its value varies according to the model. The hatch scale is controlled by the [SCALEF command](#).

Front slabs

You can disable generation of front slabs. If you decide to generate front slabs you can assign them with their own slope. It is also possible to choose the width of the front gutters or the elevation of the front gutters instead.

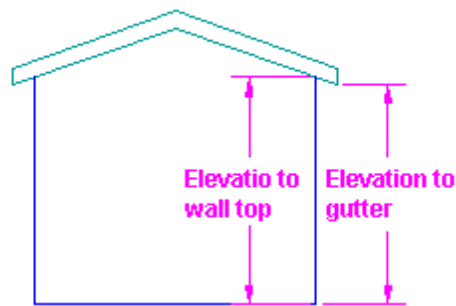
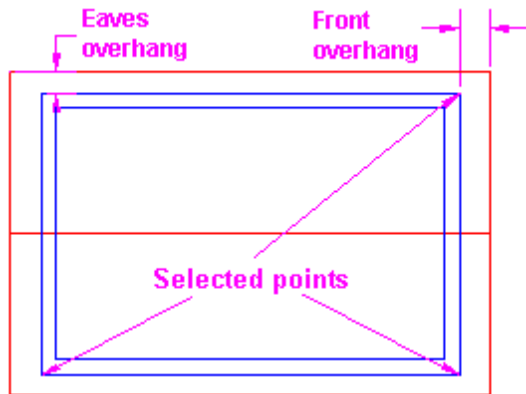
Once the generation parameters have been assigned and the dialog box closed, you must

indicate the three characteristic roof points. The first two identify a roof gutter side and the third point the other gutter side. In fact the request messages are:

First endpoint of the gutter side:

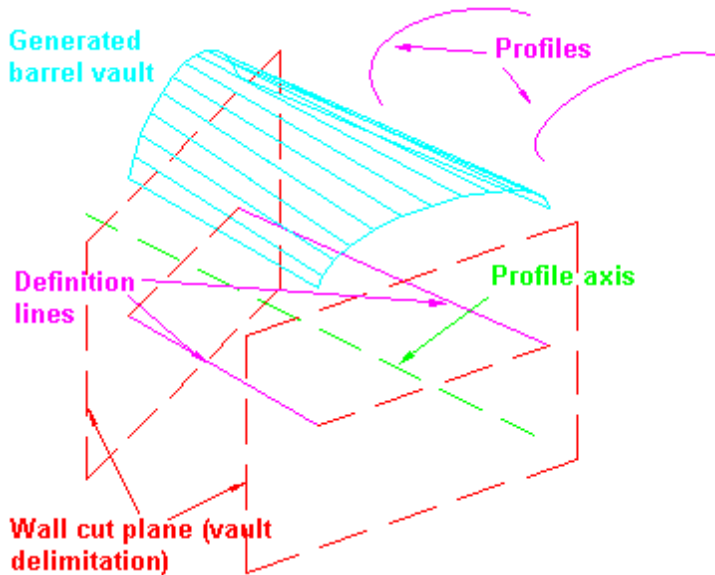
Second endpoint of the gutter side:

Select a point on the opposite side:



Barrel vaults

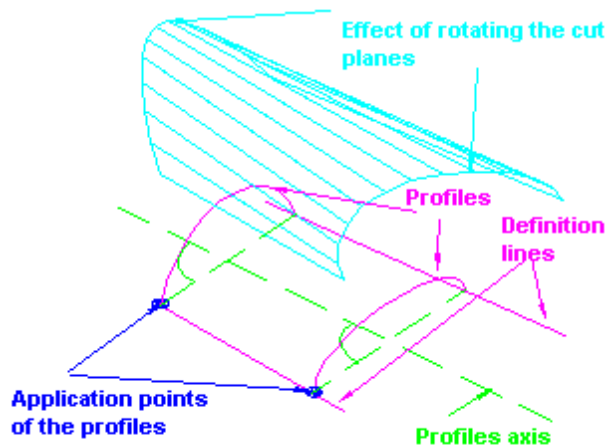
The command presented on this page makes it possible to construct rather complex barrel vaults and curved roofs. The difference between a curved roof and a simple vault is whether or not there is roof thickness.



A curved wall is defined by selecting two definition lines and one or two profile curves. The first definition line also identifies the generation elevation of the vault and whether or not it is a flying buttress, based on whether or not the elevation of the endpoints are at different heights. The z coordinates of this line are therefore very important, though it is easy to shift the vault in direction z after it is generated, as has been done in the figure. The z coordinates of the second definition line are ignored. Its endpoints together with the endpoints of the first definition line identify the two planes which cut the barrel vault. The cut planes can also rotate. To correctly identify the vault geometry, the generation lines cannot form an angle beyond 45°.

The profiles can be lines, arcs or ellipses. Combining these profiles allows you to obtain almost all types of barrel vaults. If you insert thickness greater than 0, it is possible to make any type of curved roof.

Other tools are used to specify the barrel vaults even better.



The two definition lines can not be parallel one with another, and so in general the profile axis is the bisection of the angle of two definition lines (purple in the figure). Normally when profiles are drawn, their dimensions are detected at a certain point of the vault. Therefore the 'application point' of each profile is requested.

We shall now analyze the command requests and we shall see how the concepts we have just explained operate.

Command: VBARREL

Thickness <0.0>:

Select the first definition line:

Select the second definition line:

Select the first profile:

Specify the point where to apply the first profile, or Enter in case of only one profile:

If you press *Enter*, the parameter insertion phase ends and the program immediately generates the vault. This is an abbreviated form to specify the type of barrel vault. It is

useful for regular plans, for example rectangular. The endpoints of the first definition line and the zero inclination angles are taken as application points of the profiles. If you choose a point, the command will ask the following information.

Rotation of the first cut plane <0.0>:

Select the second profile:

Obviously the first can be chosen.

Specify the point where to apply the second profile:

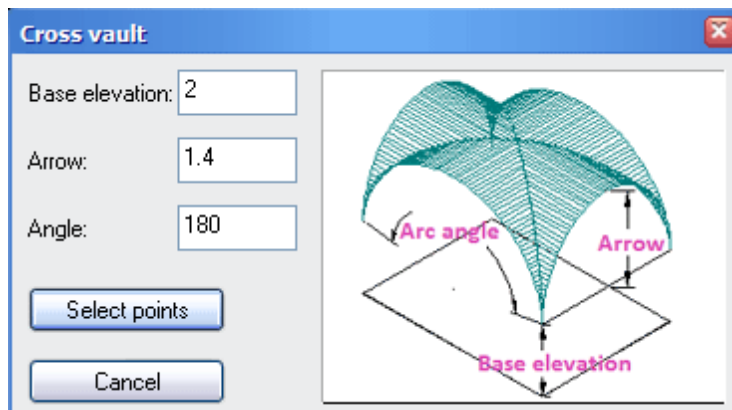
Rotation of the second cut plane <0.0>:

The entities generated are 3dface surfaces. These entities cannot be selected with the roof modification commands. The vertices can be moved by stretching the entity grips. The surfaces created save the information required to generate variable height walls. Therefore when walls are generated, the surface of the walls are aligned perfectly with the surfaces of the vault. This command uses the 3DVAULT layer. The name of the layer can be modified with the ADDCSETVAR SETBOT_LAYEREN command, or even quicker with the DDLCOLOR command, just assigning the layer to be generated with a different name.

Approximation with flat elements

AddCAD approximates profile curves with linear segments. The result is a set of flat surfaces which approximate the vault or curved roof. The number of segments to approximate the curve is a variable called SETBOT_NUMSAMP. The ADDCSETVAR SETBOT_NUMSAMP command must be used to modify the value. If for example we wish to approximate the profiles with 12 elements, we only need to write ADDCSETVAR SETBOT_NUMSAMP 12.

Cross vaults



Cross vaults typically cover square or rectangular coverings, though the program accepts quadrilateral definitions in general. The profiles can have a circular or elliptical arc. This depends on the parameters and on the dimensions of the sides of the underlying plan. If for example the angle of the arc is 180 degrees, then the arc must be half of the side to be an arc, otherwise the profile becomes an ellipse.

However the constraint of this type of vault which requires the top surfaces to be coplanar is always respected. Therefore for square plans and for certain arrow and angle values, both profiles will be the shape of a circular arc. For rectangular plans, one profile could be a circular arc and the other an elliptical arc. The command simply requests to insert the parameters of the profiles and the base elevation of the key definition lines which are the same for all. Finally the four points of the plan quadrilateral are selected to conclude the command and to generate the surfaces of the vault.

The entities generated are 3dface surfaces. These entities cannot be selected with the roof modification commands. The surfaces created save the information required to generate variable height walls. Therefore when walls are generated, the surface of the walls are aligned perfectly with the surfaces of the vault. This command uses the 3DVAULT layer. The name of the layer can be modified with the ADDCSETVAR SETBOT_LAYEREN command, or even quicker with the LAYERC command, just assigning the layer to be generated with a different name.

Approximation with flat elements

AddCAD approximates profile curves with linear segments. The result is a set of flat surfaces which approximate the vault. The number of segments for approximation of the curve depends on Approximation of curved elements the value of which can be modified with the [SCALEF command](#). To be exact, the width of the approximating face is practically half the value of *Approximation of curved elements*.

Hatching roof slabs

The HROOF command automatically creates the hatch of the selected roof slabs and floor slabs. The screen which the command chooses automatically depends on the slope of the selected element. If there is no slope, then HROOF applies the hatch model for floor slabs, otherwise it applies the one for roofs. Therefore the hatch model used can either be the one specified in the roof slab command dialog box or that of the FSLAB command.

A sophisticated system for hatch association to roofs and walls has been implemented in AddCAD. The system does not use the AutoCAD associative hatch, also due to the fact that it was too limited to meet our specific goals. When there are roof hatches and AddCAD commands are used which modify the conditions of the roofs in some way, it does not matter whether we are modifying a slope, inserting a gable or anything else. They are continually updated.

Roof girders

The insertion of girders helps to represent wooden roofs or pergolas.

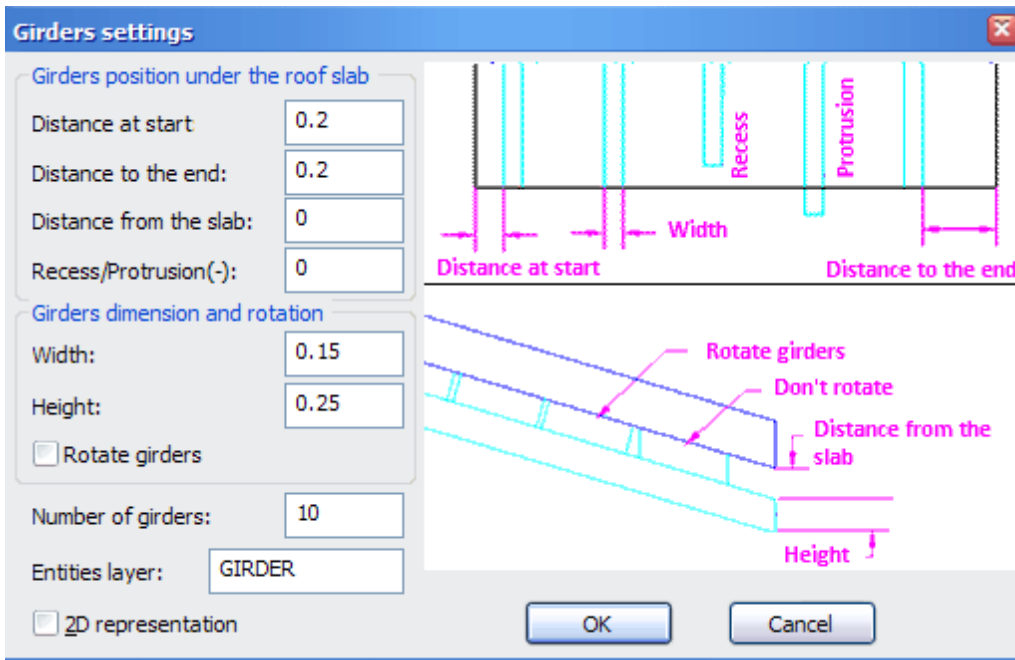
The command processes one slab at a time and then requests to select the concerned roof slab.

The side of the roof slab selected is fundamental to let the program know the direction of the girders. In fact it will always be perpendicular to the selected side. Pay attention where the roof slab is selected. According to the closest vertex, the first and last girder are chosen due to the relative distances from the vertices.

The command makes the following request:

Command: GIRDER

Select the edge of a slab that will be perpendicular to the girders or [Options/Angle]:



If you wish to set the girder generation parameters, just select *Options*. The dialog box shown in the figure will open up. The image in the dialog box explains the meaning of the parameters. *Girders position under the roof slab*

Two distinct values can be assigned for the distance of the first and last girder. The distance of the roof slab can be set at zero if you want the girders to be attached directly to the roof slab, or else at a distance equal to the thickness of the boards if present. You can use the parameter *Recess/Protrusion* to adjust the final position of the girders compared to the edge of the roof.

Girders dimension and rotation

If you choose to rotate the elements, its section will lie below the roof slab. This is the case of boards. In all other cases, it is better to choose not to rotate.

Number of girders

The number of girders determines the center distance between them.

Entities layer

Another layer can be chosen for the entities. Remember that, unlike roofs, only one layer is used for these entities.

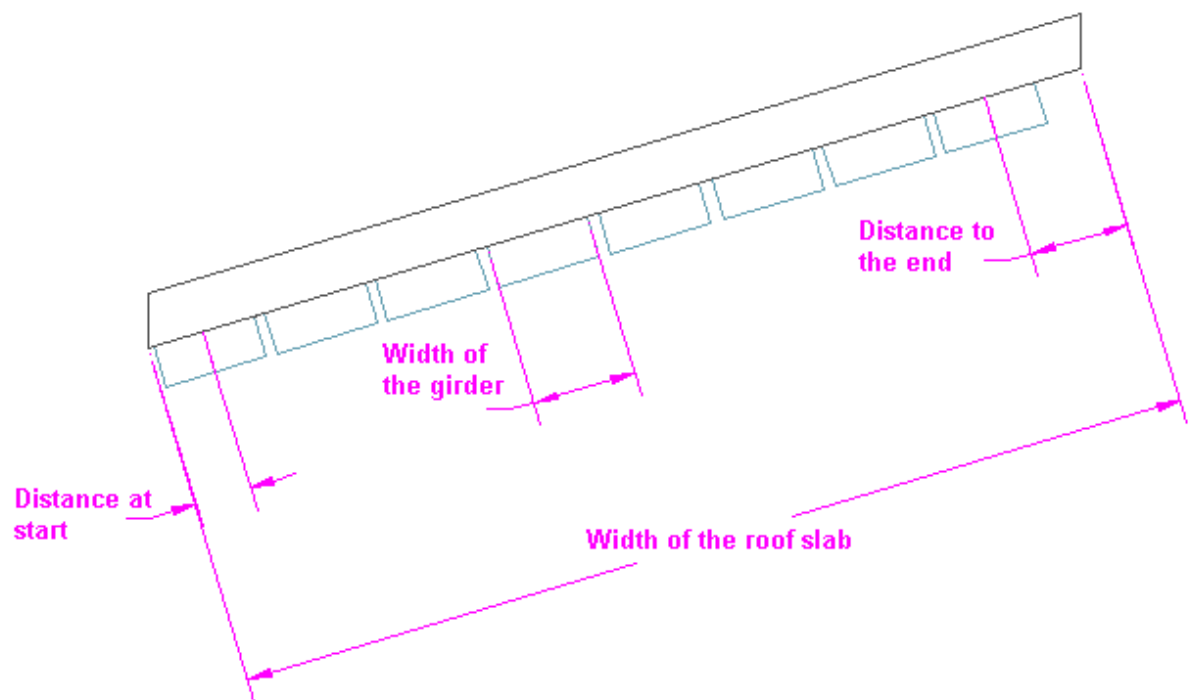
2D representation

The lines representing the girders can be generated in 2D. The setting will not be saved. Every time you use the command you must choose whether to generate the girders in 2D. You may also intervene with [girder regeneration](#) to remove or add the 2D representation. The individual girders are objects which have the same extended information as the roof slabs. This means that you can modify on them individually with the vertices modification commands and that the generation of variable height walls takes their presence into account by leaving the required recess.

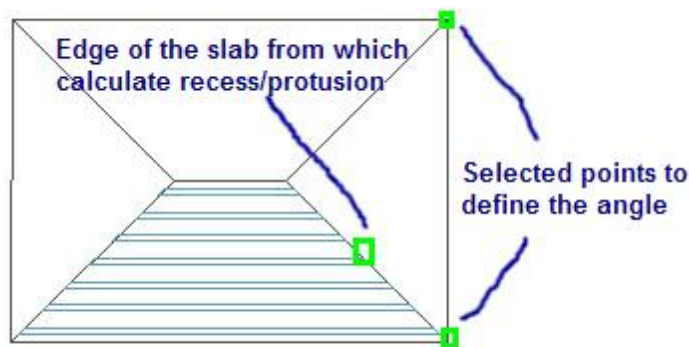
The GIRDER command proposes a standard distribution of the girders. This means that their position is normally all right, but there are times when you wish to assign the positions of the girders differently. In this case the solution is to move and copy the girders as you wish and to use the [RGIRDER command](#) to adapt them to the roof slabs.

Meaning of parameters in case of girder rotation

When rotating girders, you must pay attention to the size of the roof slab soffit and not to its casting on the floor.



Generation of girders with Angle option



The *Angle* option allows you to generate girders with a free angle. The girders will no longer be generated perpendicular to the edge of the selected roof. This option is especially useful for hip roofs.

The following are the command requests and an image illustrating an example of the selected points is shown.

Command: GIRDER

Select the edge of a slab that will be perpendicular to the girders or [Options/Angle]:
Angle

Specify the angle perpendicular to the girders:

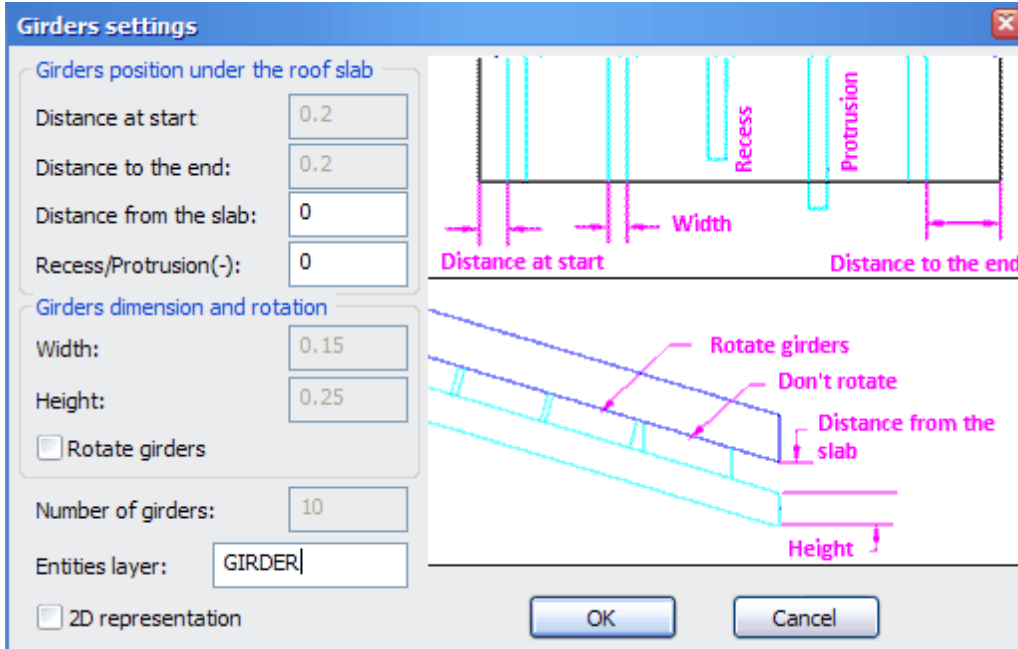
Second point:

Select the edge of a slab from which calculate recess/protusion or [Options]:

Girder regeneration

The GIRDER command proposes a standard distribution of the girders. This means that their position is normally all ok, but there are times when you wish to assign the positions

of the girders differently.



In these cases, the RGIRDER command allows you to adapt the girders to the selected roof slab. The command opens the dialog box shown in the figure and allows you to modify only those parameters regarding

regeneration of the girders in their position.

You are given the possibility of changing the distance of the elements from the bottom of the roof slab, whether or not to rotate the girder selected and to change the entities layer. You can also adjust the length of the girders compared to the edge of the roof slab. After you have exited the dialog box, the command asks you to select the roof slab and girders to be updated.

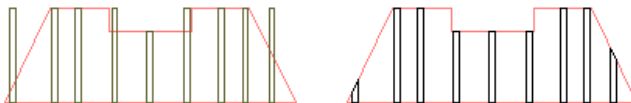
Command: *RGIRDER*

<dialog box>

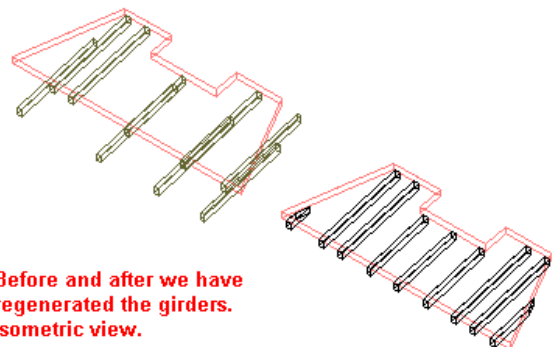
Select the edge of the slab where calculate recess/protrusion of girders:

Select the girders you want update:

The example in the figure shows the possibilities of this command.



Before and after we have regenerated the girders. Top view.

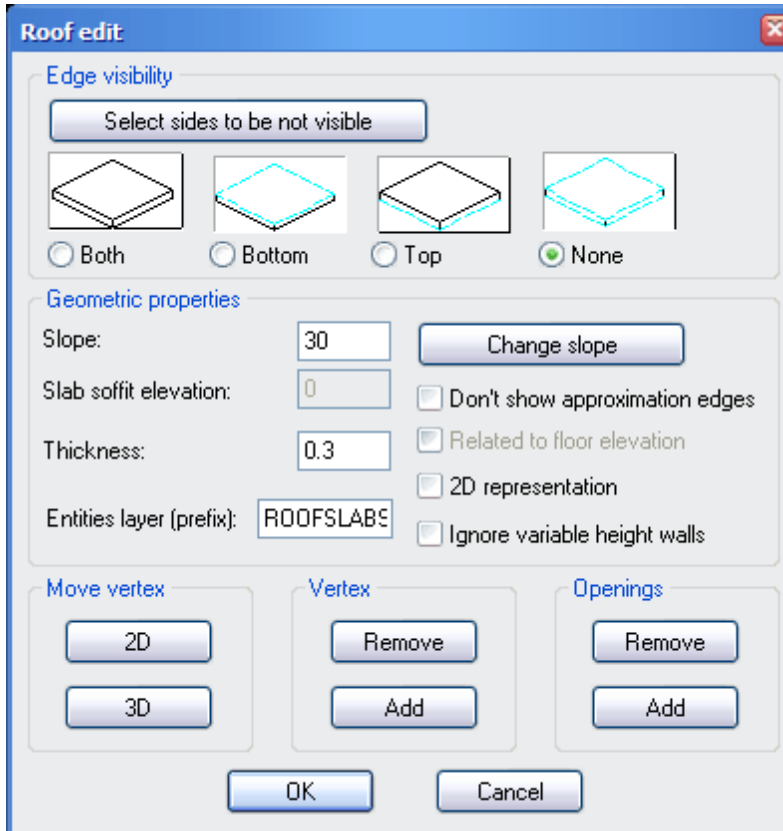


Before and after we have regenerated the girders. Isometric view.

Roof editing

Roof edit

The EDITSLABS command allows you to change many features of the roof or floor slabs. EDITSLABS modifies the attributes and geometry of the roof slabs and is capable of fully transforming its initial drawing. This command works either with sloped roof slabs or floor slabs and demonstrates the fundamental equivalence between roof elements.



After having selected the roof, the dialog box is displayed with all the current roof attribute values.

The subsequent requests of the command will depend on the type of modification you wish to make.

If you choose to modify the constructive properties and not the geometry of the roof, the command will make no more requests. By *Geometric properties*, we mean the visibility of the horizontal and vertical edges, the thickness, the generation of the 2D representation, the slab soffit elevation, relation with the floor elevation, properties respect to the variable height walls and the entities layers.

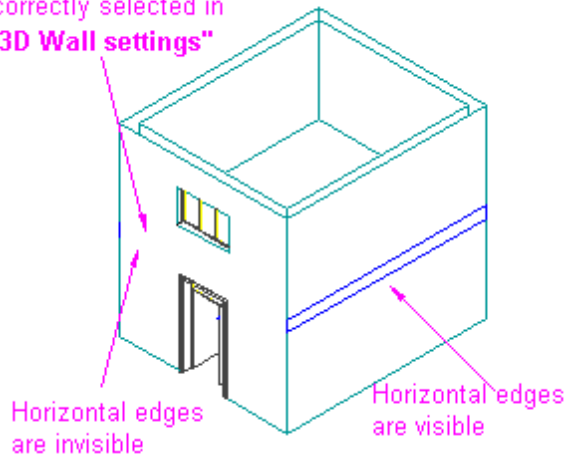
Speaking of layers, remember that only the common part of

the layers used can be modified in the dialog box. If for example we name the entities layer FLOORSLAB, AddCAD will still generate different layers. For example for the floor slab 3D, it generates FLOORSLAB_T, FLOORSLAB_B, FLOORSLAB_S, where the Top floor surfaces, Bottom floor surfaces and Side surfaces are placed respectively. This allows you to match the various layers upon which the various 3dface surfaces are placed with different colors and materials.

Edge visibility

The possibility of changing visibility of the horizontal edges is very useful. This function is required every time you wish to change the visibility of floor marking lines or whenever there are several slabs close to each other with different thicknesses. By pressing *Select sides to be not visible* you may also intervene on some sides of the roof and not on the whole perimeter. The next figure shows the assembly of two floors and a middle floor slab with the floor marking visible on one side alone.

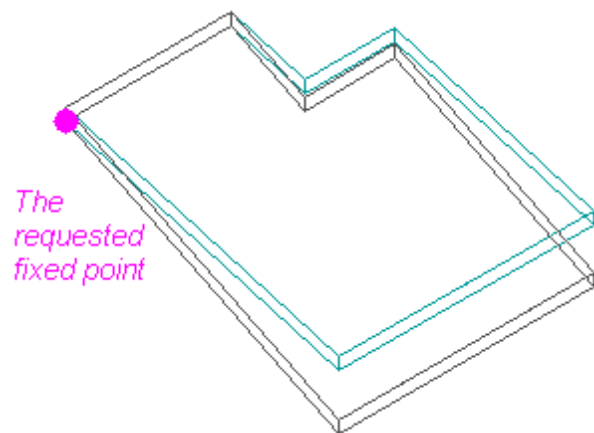
The wall edge visibility are correctly selected in "3D Wall settings"



Ignore variable height walls

When variable height walls are generated, the 3D wall stops below the roof slabs or floor slab. When this option is enabled, the presence of the roof element will no longer have effect on 3D variable height wall generation.

Once the type of modification has been chosen, further indications are required concerning the roof vertices to change roof slab geometry. By *roof slab geometry* we mean its slope, the position of the vertices and its holes.



Change slope

If a horizontal roof element is chosen, the slab soffit elevation is displayed. If you choose an inclined roof slab, the Roof slope value appears. If you choose to change the slope, the command requests to indicate a fixed point. This point will be the roof slab point which will not be moved. The point indicated must be a roof slab vertex.

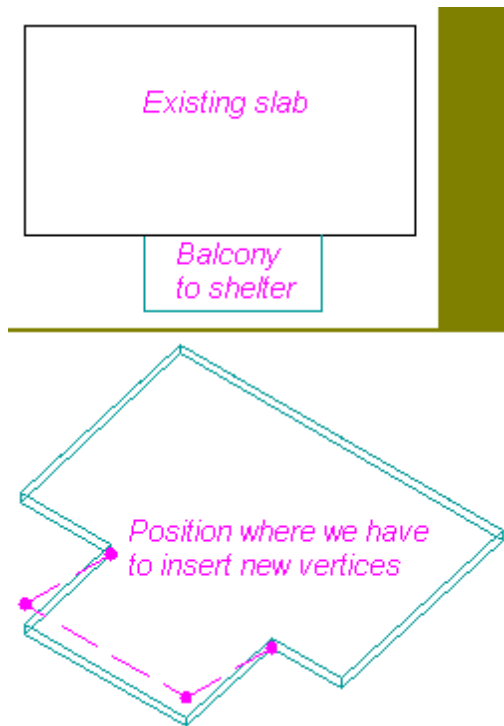
Add and remove openings

It is possible for example to open a floor slab by selecting the underlying staircase even after having constructed the floor slab. As usual, you are asked the roof element you wish to change and, after you have exited the dialog box, you are requested to perform the opening: selecting an object, selecting a closed contour and indicating a certain number of opening vertices.

You can close an opening by selecting a vertex which is part of the opening.

Add and remove vertex

If you choose to eliminate vertices, the command asks you to select which ones. The vertices can either be on the external contour or on any internal openings. If less than three vertices remain on an opening, it is completely closed. In order to add vertices, you are asked to select a side where to insert a vertex and then the position of the vertex. Let's imagine that we have a rectangular roof slab and we wish to create a slope on the gutter side in order to cover the balcony below. Four vertices need to be inserted. A great help for positioning the vertices are the lines which represent the area to be covered. After having closed the dialog box by pressing *Add Vertices*, the requests are the following.

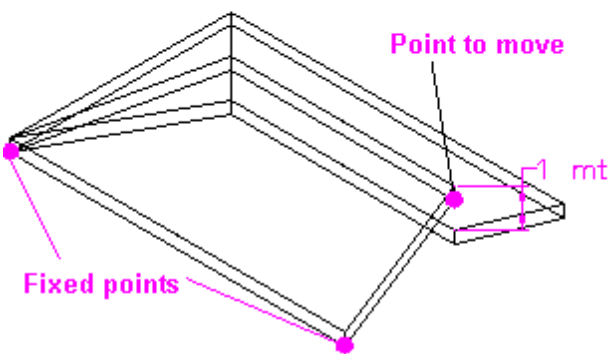
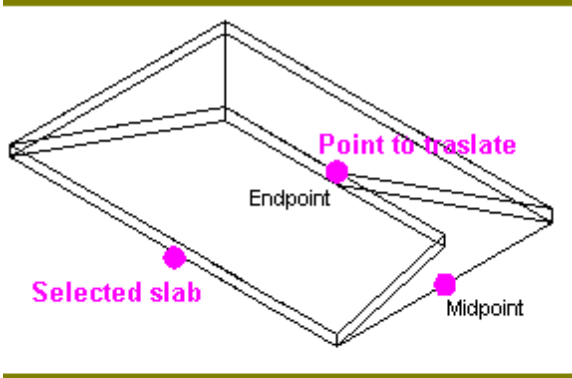
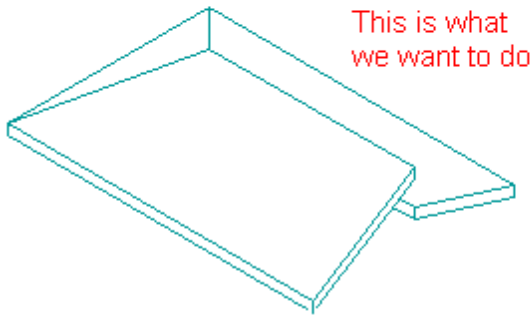
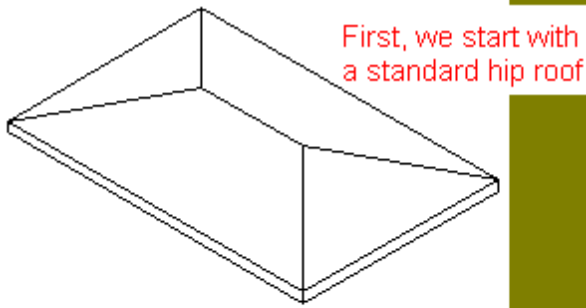


Selezionare il lato dove si vuole inserire un nuovo vertice: bottom side selection

Posizione del nuovo punto: EndPoint of corner

....and so on for all four vertices....

Selezionare il lato dove si vuole inserire un nuovo vertice: Enter to end



Last point to move: Ridge point 3

Indicate the start point: 0,0,0

Indicate the end point: 0,0,1

The movement vector can also be given by indicating the coordinates of the two points.

Move 2D vertices

This type of modification makes it possible to stretch the vertices on the xy plane, either if they are internal contour vertices or external polygon vertices. The slope of the roof slab remains unchanged. The command asks you to select all the vertices to be stretched and then requests the movement of the vector in the form of two points, the z coordinates of which are ignored. If you need work a lot with roof slab vertices, we recommend using the [specific EDITROOFP](#) command which allows you to move and add vertices in a very quick way.

Move 3D vertices

In practice this is a redefinition of the roof slab plane. You are requested three points. Generally this option modifies the slope of the roof slab. The first point is always fixed and does not move, the second can either be kept still or can be moved together with the third. In the example, wishing to move the ridge of one of the two roof slabs up one meter, we need to do this modification.

Example

After having removed the triangular roof slab on the right, we must press *Move vertex 2D* twice and then press on one roof slab *Move vertex 3D*.

Move 2D vertices

Select a point to move: Osnap endpoint

Select a point to move: Enter because only one

First movement point: Endpoint

Second movement point: midpoint

Repeat to modify second roof slab.

Move 3D vertices

First fixed point: Gutter point 1

Second fixed point/Enter: Gutter point 2

Intersection points on roof slabs

This command allows you for example to solve the problem of intersections between roof slab systems.

Thinking in general, this command allows you to find one or more points on the bottom surface of a roof slab resulting from intersections of straight lines with the roof slab. The straight lines must be indicated by selecting pairs of points which do not coincide.

For better use of the points found, the program draws a breakage between the various points which are cut bit by bit. If you need to find only one point, you must indicate the same straight line twice by means of the same couple of points. The program will draw a line at length zero at the point found.

The command follows the next path of requests.

Command: FPSLAB

Select the roof slab on which to project the points and draw the lines:

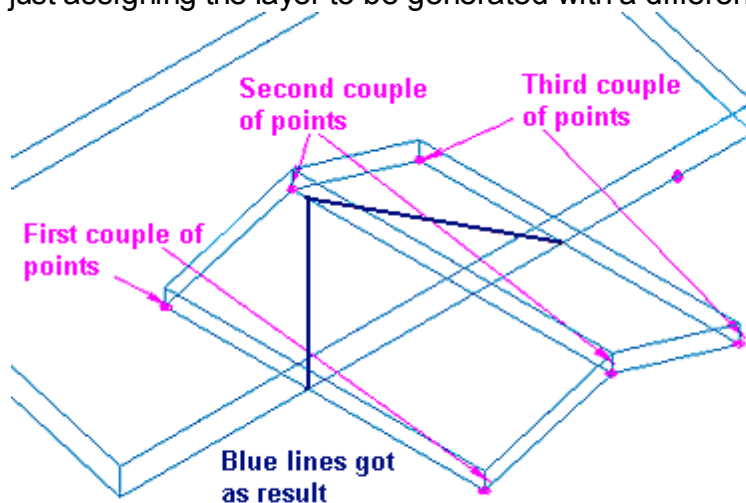
Then the following two requests are made cyclically. Remember that at least two pairs of points must be indicated.

Specify the first point:

Specify the second point:

The lines generated are placed on the HELPLINES layer, and therefore they can be deleted using the [LAYERDEL command](#).

The name of the layer can be modified with the ADDCSETVAR SETCOP_LAYERCOSTR command, or even quicker with the [DDL COLOR command](#), just assigning the layer to be generated with a different name.



Manage roof vertices

This is a command which greatly speeds up roof editing operations, especially concerning the insertion of new vertices and moving existing vertices. The same operations can be done with the more general [roof edit command](#). However this command is much quicker when modifying the vertices.

The insertion of new vertices on a roof slab and moving existing vertices does not change the roof slab plane and therefore the slope of the roof slab. When positioning points with the cursor, only x and y coordinates are considered. The z coordinate is calculated by the command so that it coincides with the roof slab plane. The command starts by requesting to select a roof slab and gives the following options:

Command: EDITROOFP

Select a slab or a profile:

The program makes the following request cyclically:

Select a point to move, Enter to exit or[Change slab/Insert vertex]:

If no points are selected, press *Enter* and exit the program. If points are selected, it asks the movement vector. It moves the vertices and again requests new points to be moved.

Specify the start point of the translation:

Specify the end point of the translation:

The *Change slab* option allows you to change the roof slab to be modified without exiting the command.

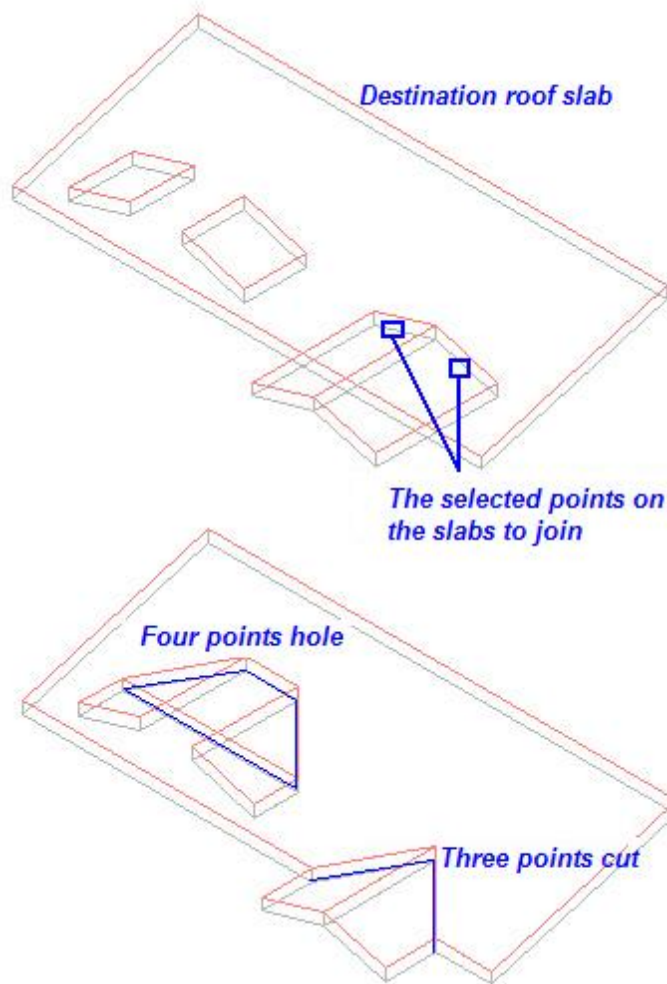
The *Insert vertex* option changes how the command works, cyclically making the following requests.

Select the edge in which to insert a new vertex:

Specify the position of the new point:

As you can see, the roof slabs are modified considerably in just a few seconds.

Joining three roof slabs



The J3SLABS command solves the recurrent problem of intersecting roof slabs to make a gable or similar roof shapes. The system based on the commands to [find points](#) on roof slabs and [move and add vertices](#) is very flexible and powerful but takes more time when it could be done automatically with the J3SLABS command. The command asks you to select three roof slabs.

The first is the one upon which the other two will be cast. Depending on the elevation of the two roof slabs, we can obtain a hole or cut in the first destination slab. Furthermore if the two roof slabs do not form a common summit (if detached one from another) there will be a hole or cut formed by four vertices instead of three. The image shows you what we are talking about.

The command makes the following requests.

Command: J3SLABS

Select the destination roof slab:

Select the edge of the first slab you want to cast:

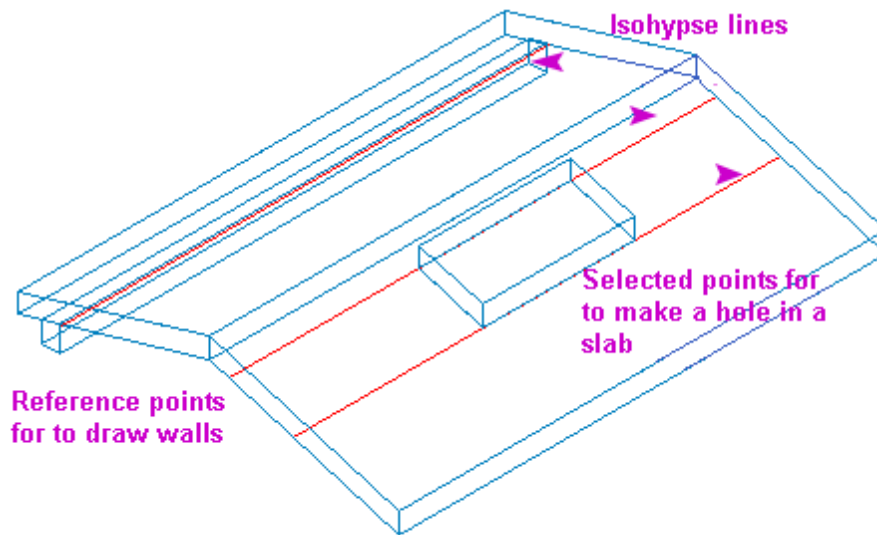
Select the edge of the second slab

you want to cast:

Important notice concerning selection

When two roof slabs are selected to be cast, it must be done on the lower edge of the side you wish to cast.

Isohypse lines



This command is very useful when you wish to modify floor slabs whose design constraints depend on elevation parameters.

For example, when you wish to build a wall at a certain height, you must place a hole on the floor slab between one z elevation and another. The ROOFL command draws lines at the requested z elevation. The requested z elevation is relative to

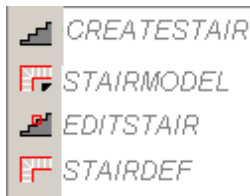
the floor elevation of the current floor. It corresponds to the absolute z elevation of the UCS World only when working at zero floor elevation.

The lines generated are placed on the HELPLINES layer, and therefore they can be deleted using the DELAYER command.

The name of the layer can be modified with the ADDCSETVAR SETCOP_LAYERCOSTR command, or even quicker with the DDLCOLOR command, just assigning the layer to be generated with a different name.

Stair modeler

The drawing of staircases has been enhanced with a stair modeler capable of drawing stairs with extreme flexibility and complexity. The [parametric approach](#) greatly simplifies the process for those standard situations which can be resolved with parametric objects.

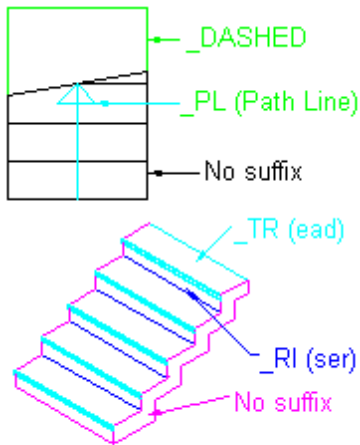


Drawing stairs with the modeler fundamentally starts from what we'll call the [definition model](#). The definition model consists in line and arc entities which describe the 2D plan shape of the stairs. The model can be generated by hand by using AutoCAD to draw lines and arcs, or else using a specific command called [STAIRDEF](#). The first method is undoubtedly more flexible while the second is faster.

The STAIRDEF command has an option which allows you to generate the stairs right away or it draws the entities of the definition model and allows you to intervene manually on the model generated automatically with AutoCAD for greater flexibility.

The [CREATESTAIR command](#) is for creating stairs from a definition model. After having selected the model entities, the command opens a dialog box where you may choose the type and dimensional parameters of the staircase. When values have been assigned to the parameters, the command immediately generates the stairs in all its details.

It is possible to modify an existing staircase. The [EDITSTAIR command](#) allows you to modify the parameters and type of stairs. The [STAIRMODEL command](#) allows you to transform the stairs to pass onto their initial definition model.



Stair layers

The layers which the stair modeler uses can be modified in [Parametric and stairs layer tab](#) of the ADDOPTIONS command. We recommend only changing layer names for specific needs. The layer generation attributes should be modified with the [LAYERC command](#).

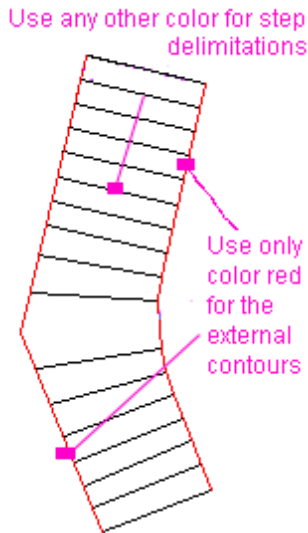
Stairs and other AddCAD features

Stairs can be selected with the definition and edit roof commands to generate a hole in the floor slab. The size of the hole depends on the headroom height set while creating the stairs.

The area below the stairs can be deducted from the room floor calculation. The area depends on the parameter *Deduction of stair projections* specified in [Quantities survey system tab](#). The stairs also represent a hatch island for the floor hatch command.

Handrails can be applied to the stairs with or without railings.

Definition model



Drawing stairs with the modeler fundamentally starts from what we'll call the definition model. The definition model consists in line and arc entities which describe the 2D plan shape of the stairs. More precisely, we distinguish between stair contour and steps.

OK. AddCAD extends the lines automatically

Error.
Use only red color for external contours

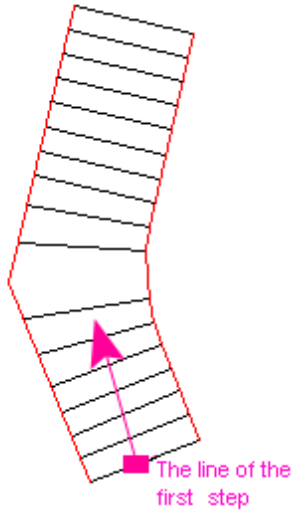
Error.
The first step must start and end on the external contours

The contour must be composed of two pairs of arcs and line sequences. The delimitations of steps can only be lines. The goal of speeding up the selection of model entities has led to making rules which must be respected in the model definition.

- The entities defining the contour must be red.
- The lines delimiting the steps can be any color but red.
- The line of the first step must end on the two contour endpoints of the stairs.

For all the delimitations, except the first, the program calculates the intersections by which even an imperfect drawing leads to a correct recognition of the stairs.

Creating stairs from model



The CREATESTAIR command starts from a [definition model](#) and generates a staircase. Once the command has been launched, two requests are made:

Select the line of the first step:

Select the objects which define the stairs:

The line of the first step must be selected with the selection crosshair. This determines the ascent direction of the staircase. The second selection must include all the entities of the stair model.

The presence of entities not belonging to the model could trigger error messages or an incorrect generation of the staircase. The program automatically discards entities which are not lines and arcs representing walls.

If the stair definition is correct, the program immediately presents the dialog box to insert the generation parameters. Keep in mind that many of the parameters of the dialog box

propose the value of the last stair generation.

The indications in the dialog box and the help image make the parameters easy to understand. Nonetheless we will give a detailed description on this page

Stair height The Riser count is displayed

. This value is taken from the definition model and is only for a check. The option *Last tread on arrival elevation* means that the last tread will be at z elevation equal to the elevation of the floor it finishes at. If not selected, the last tread will be the extrados of the floor slab and the last tread of the stairs will be below the extrados of the floor slab by a value equal to a riser. From what we have said, it is evident that if the option is selected, there will be one less riser.

The *Start offset* value is added to the starting floor elevation to copy the starting absolute z

elevation of the stairs.

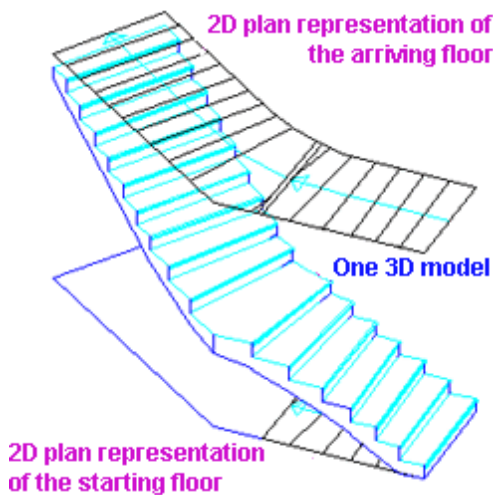
You may choose whether to specify the riser of the steps by setting a *Distance between floors* which is then equally distributed on the step risers, or by directly specifying the *Riser value*. In both cases, another control value is displayed.

Nosing length and thickness

It is possible to choose between mitered, commonly used in new constructions, and rounded, which describes a semicircle. In any event, you can intervene on nosing dimensions and put it to zero when needed.

Slab hole

During [creation of floor slabs](#), the stairs can be selected to make a hole in the slab for access from one floor to the next. The size of the hole is calculated according to the values entered in this viewport. A polygon is calculated which leaves a specified *Headroom height*. The *Slab thickness* is necessary because the height is intended to start from the slab soffit.



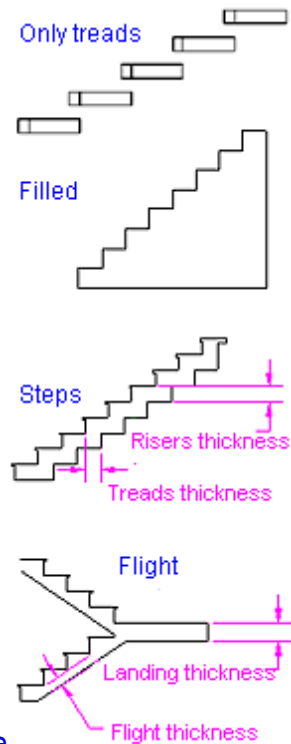
Starting from and Arriving to

The *Starting from* parameter represents the floor elevation where the stairs are inserted. [Belonging to the floor](#) for the visibility of the 3D and 2D plan models and for the insertion z elevation are decided with this parameter.

The value entered in *Arriving to* is taken into consideration only if different from that of *Starting from*. In this case, a second 2D representation of the stairs is generated, the visibility and drawing elevation of which are associated to the floor specified in arrival elevation. This allows you to have two 2D representations for the same staircase. You may also choose which type of representation to draw for the two floors.

Cut Plane height

For the 2D stairs cut plane drawing, it is possible to choose the height at which the line breaks the stairs. This parameter is quite useful when, for stair model purposes, the cut plane line falls in positions in which the drawing would not be correct, for example if the line ends up on a landing.



Kind of structure

You can define the type of stairs you wish to generate in this area of the dialog box. You may choose between filled, steps, flight or only treads. The figure shows the meaning of these options and the relative structural parameters. If you select the flight model, the *Select landings* button is enabled. If this button is pressed, the dialog box disappears temporarily and the areas considered landings can be indicated on the screen. The selected areas are highlighted in green.

Advanced stair settings

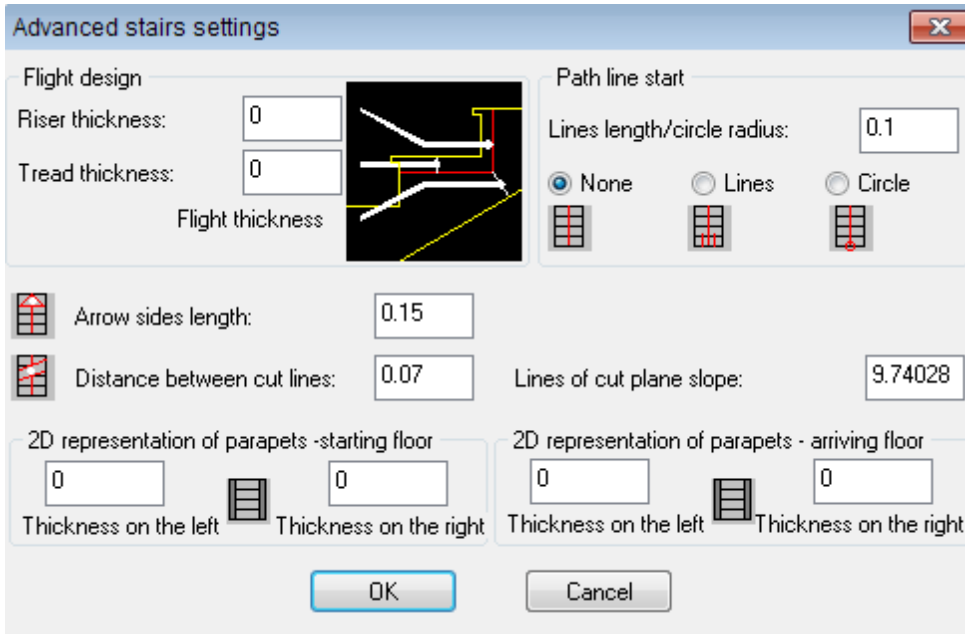
Flight design

For the flight structure, two parameters can be specified to correctly calculate flight thickness. The flight thickness can be indicated as the net value of the two settings due to the thicknesses of the stair covering.

Path line start

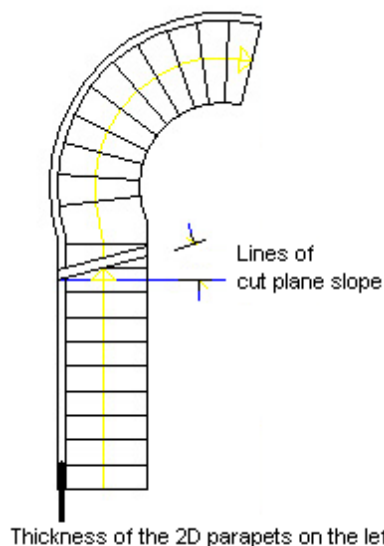
The standard symbols can be inserted at the path line start. In particular, the two small lines or the circle can be added.

Finally, the arrow symbol can be dimensioned in the distance between the cut plane lines.



Distance between cut lines

In the 2D plan representation, in overlapping mode, it represents the distance between the two cut lines of the stairs.



Arrowsides length

The side of the arrow of the path line.

Lines of cut plane slope

It is possible to vary the slope angle of the cut lines in the 2D plan representation of the stairs. The angle can be inserted in sexagesimal values.

2D representation of parapets

A handrail or parapet can be drawn along the right, left or both sides in the 2D stairs with appropriate thicknesses. If thickness is set at zero, no double line overlapping the side of the stairs will be drawn.

Nel caso di rappresentazione in pianta delle scale in modalità sovrapposte rappresenta la distanza tra le due linee di sezione della scala.

Arrowsides length

Il lato della freccia della linea di passo.

Interactive definition of stair models

This command quickly provides you with the definition model of a staircase.

The program has a series of options which, if used correctly, allow you to completely define the stair definition model, or at least to have a good draft, by selecting the [CREATESTAIR command](#).

By selecting the *Create* option, you can pass directly to the dialog box to enter the stair parameter values and then to generate the staircase.

Command: STAIRDEF

[Create/Width/Tread/Alignment/snap ON/Landing/Undo]From point:Width

The *Width* option is used to vary the width of the individual sections of the stairs.

Newvalue for stair width<1.10>:2

[Create/Width/Tread/Alignment/snap ON/Landing/Undo]From point:Tread

The *Tread* option is used to vary the tread of the individual sections of the stairs. The tread drawing can also be disabled for a section of the stairs by selecting the *Landing* option and reactivated with the *Flight* option..

Newvalue for stair tread<0.30>:.4

[Create/Width/Tread/Alignment/snap ON/Landing/Undo]From point:Alignment

The *Alignment* option allows you to decide whether to give the points on the right, on the left or in the center of the stairs.

Alignment point[Left/Center/Right]<Center>:Left

[Create/Width/Tread/Alignment/snap ON/Landing/Undo]From point:ON

Snap ON and *OFF* is used so that the cursor 'snaps' value equal to one tread.

[Create/Width/Tread/Alignment/snap Off/Landing/Undo]From point:

[Create/Width/Tread/Alignment/Arc/snap Off/Landing/Undo]To point:Arc

After having selected the first point, you may choose whether to make a curved stair section with the option *Arc*. The *Arc* option proposes two methods for tracing the curved stairs, by three points or *Center* and *Final* angle. Keep in mind that the direction the stairs rotate can always be inverted with the *Reverse* option.

[Width/Tread/Alignment/Reverse/snap Off/Landing/Center/Undo]Specify second point of arc:Center

[Width/Tread/Alignment/Reverse/snap ON/Landing/3point/Undo]Angle:

[Create/Width/Tread/Alignment/Arc/snap Off/Landing/Undo]To point:Create

Once all the sections of the stairs have been drawn, there are two ways to exit the command, other than pressing *Undo*. The first is by pressing *Enter*. This way the command generates the stair shape and riser delimitation entities, according to the lines and arcs of the definition model. It is then possible to make changes by hand and then to launch the [CREATESTAIR command](#) to generate the stairs. The second method is the *Create* option which goes on to directly create the stairs displaying the dialog box right away.

Modifying stairs

Once the stairs have been drawn, you can modify them by modifying both the structural parameters and the definition model. There are two commands which allow you to modify the stairs.

STAIRMODEL

We can say that this command 'explodes' the stairs and you go back to the [definition model](#) based on lines and arcs. The request of the command is clear:

Command: STAIRMODEL

Select the stair you want transform in a definition model:

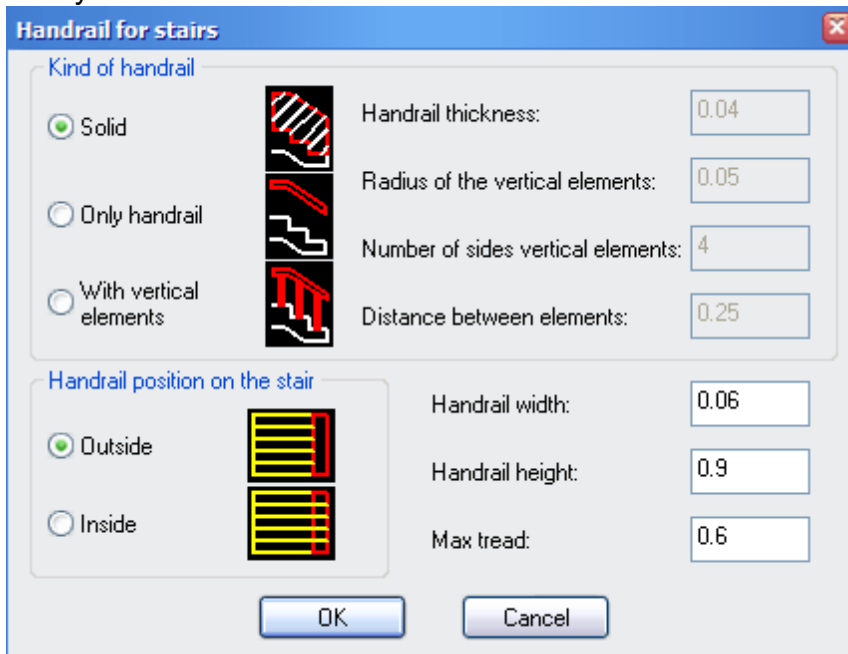
You can move riser delimitations, modify stair contours and re-create the stairs by starting over from the CREATESTAIR command. All information previously assigned in the dialog box is lost and must be chosen once again.

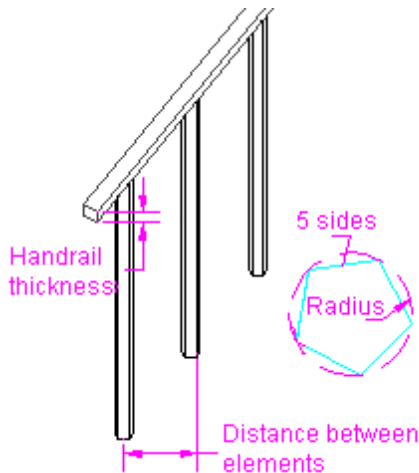
EDITSTAIR

All the parameters assigned in the [dialog box during creation](#) of the stairs can be modified. Just select the stairs and the dialog box appears with all the current parameter value.

Handrails

The HANDRAIL command automatically creates handrails on stairs regardless of whether they were created with the [stair modeler](#) or if they are [parametric stairs](#) of the object library.





Handrail creation starts from the dialog box in which you must choose the *Kind of handrail* and the associated parameters. Let's take a closer look at the contents of the dialog box

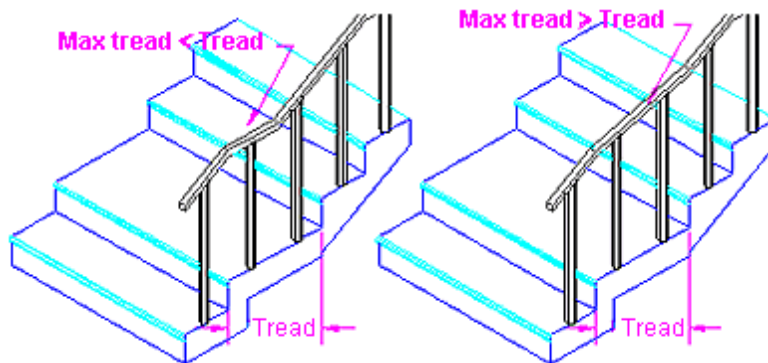
Kind of handrail

The images next to the selection buttons should describe the kind of handrail.

The parameters to the right can be enabled or disabled depending on the kind selected. No parameter is pertinent if the solid handrail is selected and therefore they are disabled.

The distance between elements only has effect if the tread or section of the landing is greater than the set value. Meaning that if the tread is smaller than the set

distance, an element is still drawn.



Handrail position on the stair

It is possible to choose whether to generate the handrail outside of the stairs at the side of the staircase, or placed on the steps, and therefore inside the staircase.

Max tread

The handrail command generates continuous handrails between the two points indicated on the stairs. This means that the handrail follows the slope of the stairs provided for each step by the riser on tread ratio. In sections where the tread exceeds a certain value, for example on landings, the handrail must be horizontal then merging with inclined sections. The *maximum tread* parameter indicates the maximum tread value by which the handrail must be created in inclined mode.

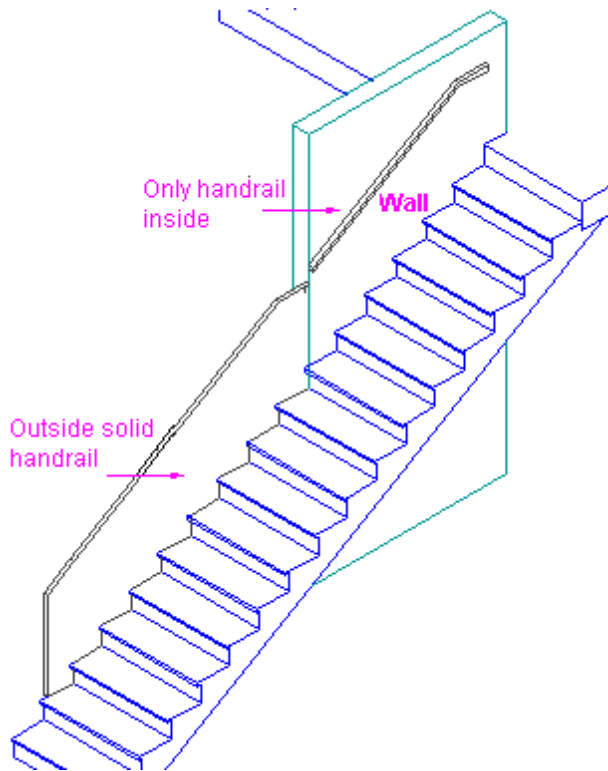
Once the kind of handrail has been chosen and the dimensional parameters assigned, you must indicate which side of the stairs the handrail is to be inserted and the interval on the stairs to be considered. Often the handrail is not generated along the whole side of the stairs. The command therefore asks:

Select the stair on the side where to insert the handrail:

Start point of the handrail or Enter for the whole side:<POINT>

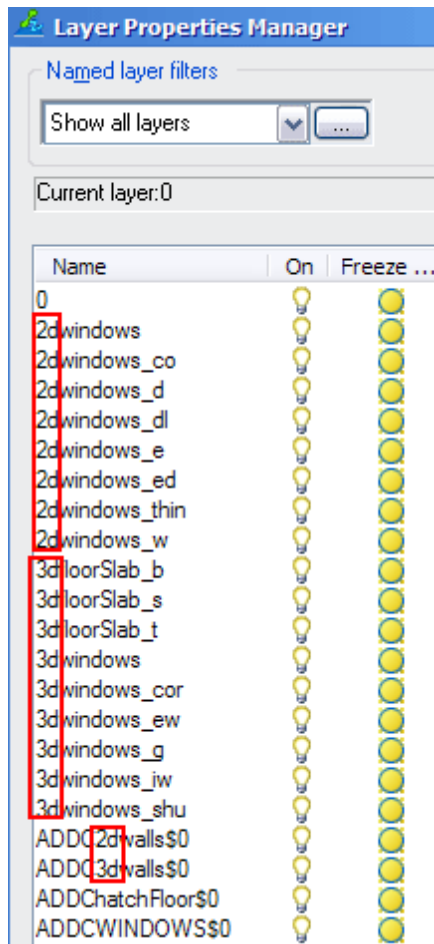
Handrail end point:

It is also possible to place two different kinds of handrails on the same side of the stairs.



The handrail is an object which cannot be modified. The speed by which it can be inserted makes this command superfluous. There are two layers where the handrail entities are inserted, one for the handrail, the other for the railing and both can be modified with the ADDOPTIONS command [Parametric and stairs layer tab](#).

2D plan and 3D views



The VXD command, by using the prefixes in the entities layer names, allows you to switch the view between layers containing entities of the 2D representation and layers containing entities of the 3D representation. After having used AddCAD for a while, you will see that the layers are arranged to separate the plan representation from the three-dimensional representation for visualization.

Command: VXD

View[2D/3D/Both]:

If you choose *Both*, all entities will be viewed, both those of the 2D and 3D representations.

If a certain viewing status is active, for example 2D, the new layers of the other representation, for example 3D, will be generated with a disabled visualization status.

Model visibility and Layout visibility

The command works correctly both in layouts and model space. In fact, for local Layout windows, the command freezes and thaws in the current window. This way the 2D plan model can be viewed in one window and the three-dimensional model in another.

Viewed representation and status bar

It is not always clear which representation you are seeing by looking at it. The active representation is also shown in the status bar, next to the current floor elevation.

Viewing roofs and floor slabs



The ROOFV and FLOORV commands allow you to enable and disable the visualization of the group of layers respectively of the roofs and floors with just one command. The commands work properly both in the model space and in the paper space, in the sense that in the paper space they enable or disable the view of the floor slabs and roofs freezing the layers in the current window.

Command ROOFV

The ROOFV command activates or deactivates visualization of roof layers. The command makes the following requests.

Command: *ROOFV*

Roof slabs [On oFf]<oFf>:

Enter or *OFF* deactivates, *ON* activates the view of the roof layers. To know the name of the layers belonging to the roof group, AddCAD looks at the contents of the VLROOFS.LST file which contains the list of the layer names. Names which can include special characters to define the names. The standard content of the file is
ROOF

with this name definition, all layer names which contain 'ROOF' are considered.

Command FLOORV

The FLOORV command activates or deactivates visualization of floor slab layers. The command makes the following requests.

Command: *FLOORV*

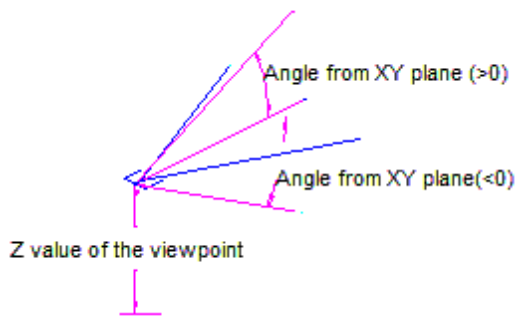
Floor slabs [On oFf]<oFf>:

Enter or *OFF* deactivates, *ON* activates the view of the floor slab layers. To know the name of the layers belonging to the floor slab group, AddCAD looks at the contents of the VLFLOORS.LST file which contains the list of the layer names. Names which can include special characters to define the names. The standard content of the file is
FLOOR

with this name definition, all layer names which contain 'FLOOR' are considered.

Perspective views

The PERSP command allows you to obtain internal and external perspectives by selecting particular parametric objects called photo cameras. The command makes a simple request to select the photo camera.



The photo camera is a standard AddCAD parametric object. It can be modified like all other parametric objects. And it is possible to insert multiple cameras in a drawing, one for each perspective viewpoint.

Visual field

The camera displays the visual field determined by an opening angle. The viewed cone is a good way of representing what will be seen whenever the camera is selected with the PERSP

command. The length of the angle lines determines the length of the help lines which indicate limits of the visual field.

Z Value of the viewpoint

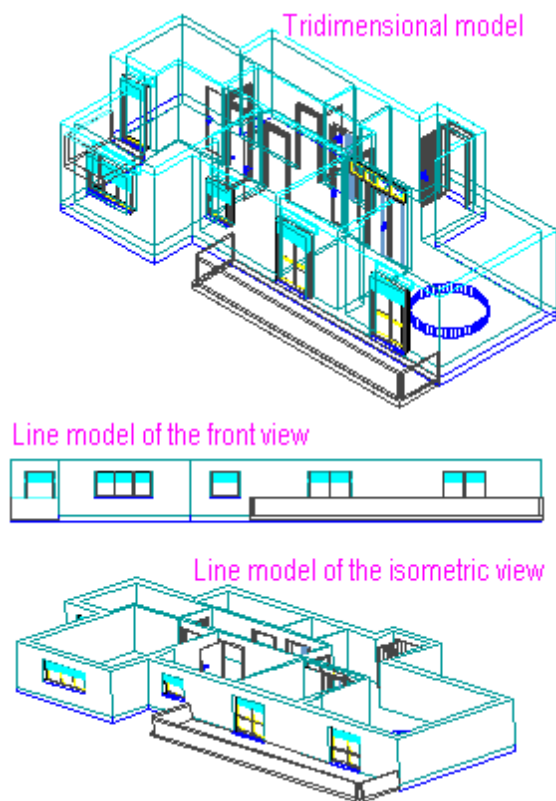
The *z value* of the viewpoint identifies the height of the viewpoint. The value is relative to the global coordinate system.

Direction of view

The parameter *Angle* with respect to the horizontal plane represents the viewpoint lowering (negative values) or raising (positive values) its focus.

Generation of views

3D views to line models



One of the main objectives for implementing the three-dimensional model is to plot of drawings with different views: plans, front views, sections, isometric and perspective views. Windows could be opened directly in the Layout, various viewpoints chosen, paper space dimensioned and plotted. This strategy is valid only if the various views do not require special interventions.

Often the 3D model is not enough to obtain complete drawings of the various views. In general, they must be dimensioned and details added which were not present in this three-dimensional model. In the end, it is useful to obtain representations based on lines of the 3D model views.

To automatically obtain sections, front views, perspective and isometric views which are also correct from a viewpoint of the architectonic process, the three-dimensional model must be accurate.

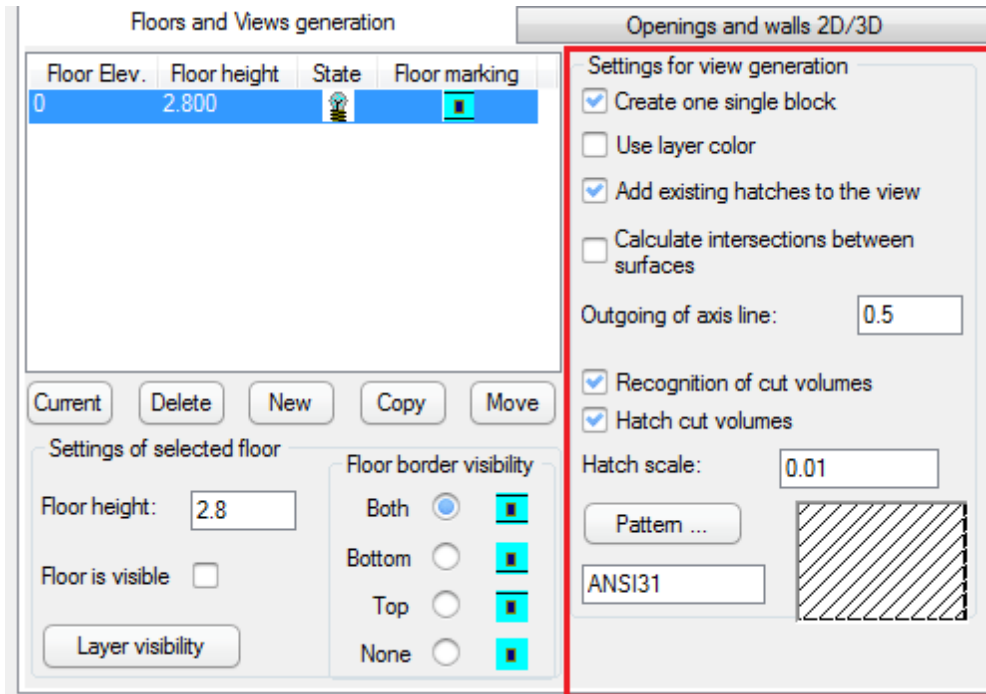
The technique which allows you to obtain representations made up of lines starting from a

3D model consists in the appropriate use of the [GENVIEW command](#).

This command manages only 3dfaces, hatches and lines on 3d layers, the entities with which AddCAD constructs 3D, but is able to represent cut surfaces and intersections between surfaces. It is also possible to recognize the cut volumes and to apply a hatch model to them. It is precise and generally fast and simple.

Line model with GENVIEW

The command allows you to generate a model of lines which represent the casting of the 3D model on a plane. The plane can cross the model, thus obtaining a section. The command places the lines on two different layers by inserting those which represent intersections between the 3dfaces and the section plane in ADDCSECTION_SEC and the others in ADDCSECTION, in order to manage them separately.



Some command options can be set in *Floors and views generation tab* of the **ADDOPTIONS** command. *Create one single block* One single block with all the lines can be generated or else the lines can be generated in the

drawing without gathering them in one block.

Use layer color

The lines can be generated with the original color of the objects or with the color of the view generation layer.

Add existing hatches to the view

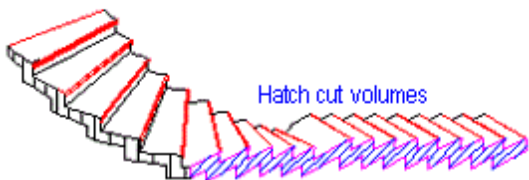
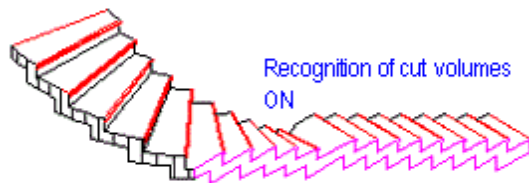
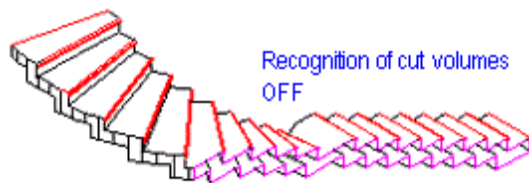
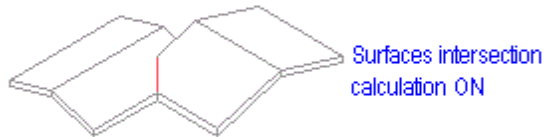
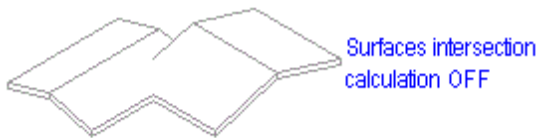
If not selected, the command only considers the 3D model made with 3dfaces. If on the other hand it is selected, even the hatches of the 3D model are processed.

Calculate intersections between surfaces

The command is capable to calculate 3dface intersections of the model, an operation however which requires more calculation time.

Outgoing of axis line

The outgoing of the line indicating the point of interruption of the broken line with respect to the top and bottom extension of the section. This parameter is valid only if the view is generated by selecting the [broken section line](#) as an object.



Recognition of cutted volumes

By enabling this option, the view generator tries to recognize the cut volumes. Actually AddCAD does not work with solids. This is why it can only 'try' to recognize the cut volumes. Recognition is obtained with an algorithm which considers the layer names and the positions of the 3D objects.

Hatch cutted volumes

By enabling this option, the view generator creates a hatch element for each area representing a recognized cut volume.

Pattern and Hatch scale

It is possible to set a scale factor for the hatch and to choose a hatch model to be used by writing the model name or selecting the *Hatch...* button which allows you to select the pattern from a new dialog box.

Layer for hatch

The default layer for hatch entities is ADDCSECTION_H. It is possible to change the attributes of this layer with the usual configuration of the layers.

Note concerning the equivalence of layer names

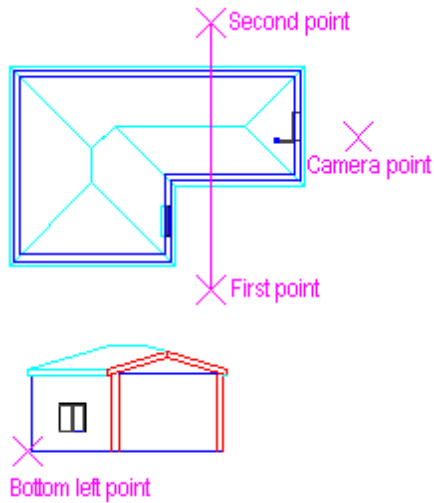
Some elements, such as floor slabs, use different layers. We have mentioned that the recognition algorithm of the cut sections also takes account of the layer names of the 3D surfaces which create lines representing their

section. In order to accommodate the two requirements, we have a file of equivalent layer names for the algorithm. A file called ADDCFLU.LST exists which considers these equivalences. The file is made up of simple lines with the following format, namely two names separated by a comma.

The names of the 3D wall layers of a floor are always deemed equivalent and therefore do not appear in the file above. More precisely, all ADDC3D*\$X files are converted into one equivalent layer.

Command line options

The options of the command line allow you to define the section or casting plane in various ways.



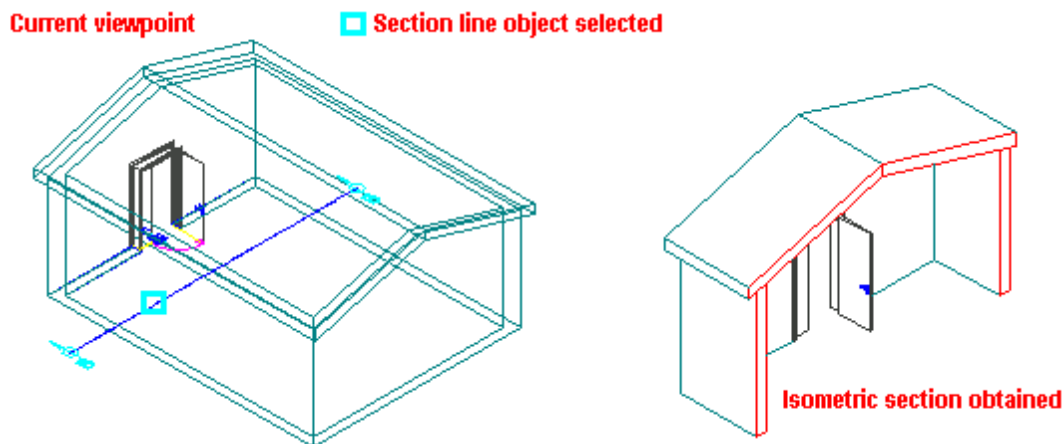
Command: GENVIEW

First point of the section line, Enter to select an object [View/Axonometric]:

The *View* option elaborates the current view. If for example you have a particular axonometric view on the screen, its model can be generated with this option.

The *Axonometric* option provides you with the section axonometric view. To be able to use this option, you must have inserted the [section object](#) provided with AddCAD in the drawing. Before launching the command, you must also place yourself at the desired viewpoint in order to obtain the section from that viewpoint.

If you press *Enter*, you can select a section object which graphically indicates a section. This method is useful to reproduce the same sections if changes are made to the project.



It is quicker to indicate points. The command requests two points where the section line should pass. Obviously if the line passes outside the building, the result will be a front view. The section line alone is not enough to define the desired view. A point is needed indicating where you wish to see the view from.

Specify the camera point:

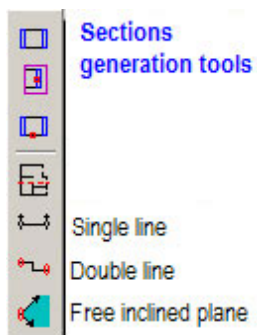
Finally the command requests to indicate the point to insert the elaborated view.

Specify the bottom left point of the elaborated view:

How to bring the views of GENVIEW in the overall layout drawing

The GENVIEW command must be run in the 3D model drawing, whichever [multi-floor design](#) has been chosen. At times the architectonic layout drawing does not correspond with the 3D drawing. You must therefore transfer the views generated with GENVIEW to the architectonic layout drawing. There are different ways to do this. If everything is done in the same drawing and plotting is set up from the Layouts, then you just need to move the sections and front views as necessary from the 3D model, in order to see only the view we want below the Layout windows. If the architectonic layout is on another drawing, then just open it and transfer the view block to other drawing with CUTCLIP and PASTCLIP.

GENVIEW with object selection



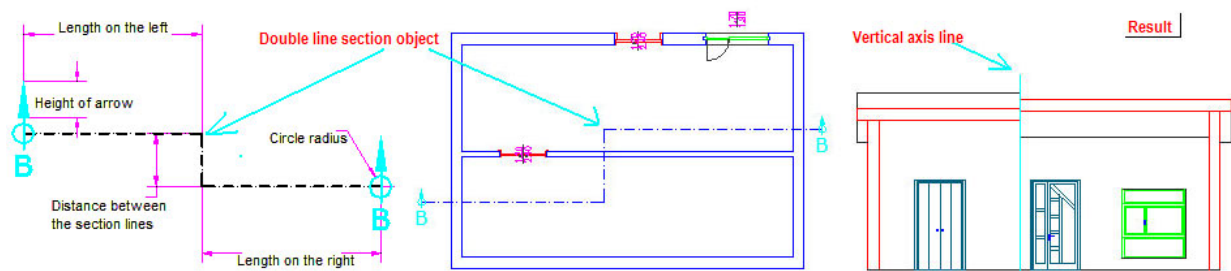
As shown on the GENVIEW command page, it is possible to select objects to generate particular views instead of and added to the simple indication of points.

Sections simbols

Three section objects can be inserted from the menu. The first, already known, is equivalent to the indication of the two points displayed, the second to obtain sections with a double line and the third to generate axonometric cutaways.

The layout of the view generation toolbar is shown.

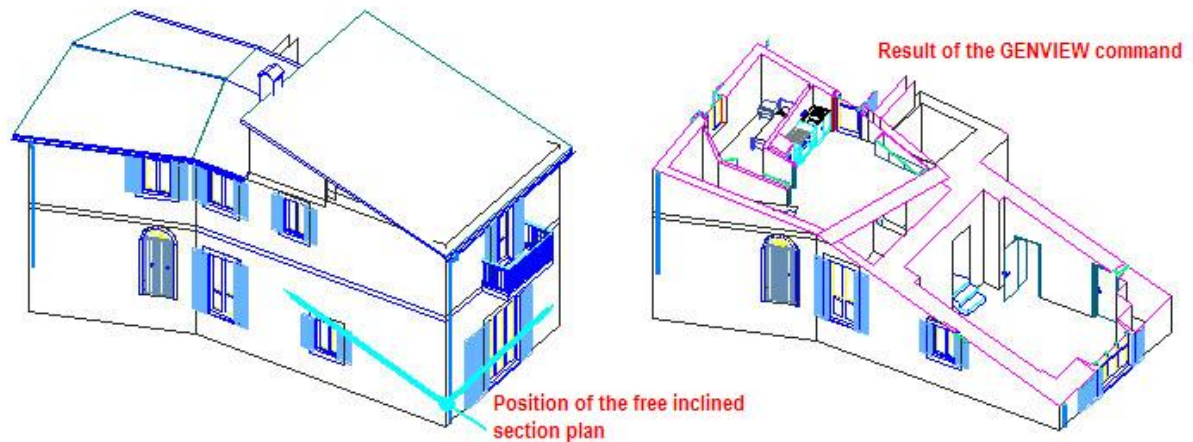
Double line section



The position of the line on the left with respect to the line on the right depends on the sign of the parameter *Distance between the section lines*. If a negative value is entered, the right line will go beyond the left line. The generation of the section will also have a vertical line at the discontinuity point. This line is generated on another layer in order to be managed separately. The layer name is ADDCSECTION_BRK. The outgoing of the line with respect to the overall encumbrance of the view can be changed by specifying the relative value in *Floors and views generation tab* of *AddCAD options*.

Axonometric cutaway

GENVIEW has a parametric object which allows you to define the section plane at will. Once this plane has been defined, you can generate the model from a viewpoint independent from the section plane. The next figure has an example of how this model is implemented.



Updating views

Once the views have been generated with the [GENVIEW command](#) or with the [GENSHADE command](#) they can be updated based on the new or changed 3D project. This command is very useful as often you need to make modifications even before the project has finished. The REGENVIEW command requests to select as many views as you wish to be recalculated. Being able to select several views to be recalculated is especially convenient, imagine a large project with several front views.

The command cannot be used with projections obtained with the object selection of the [SELVIEW command](#). In fact the viewpoint, section plane and insertion point are saved in the view block. The program is not capable of remembering objects selected whose view must be generated.

Important note concerning use of the explode command

If view blocks are exploded, obviously the above-mentioned information will be lost and therefore it is no longer possible to update the view after having exploded it.

The command asks you simply to select the views.

Command: *REGENVIEW*

Select views you want to regen:

Generating views of objects

The SELVIEW command allows you to generate the 2D casting of the views by selecting only the desired objects. Once the command has been launched, the dialog with the user is the following.

Command: *SELVIEW*

First point of the section line, Enter to select an object [View/Axonometric]:

...choice of option or indication of points...

Specify the bottom left point of the elaborated view:

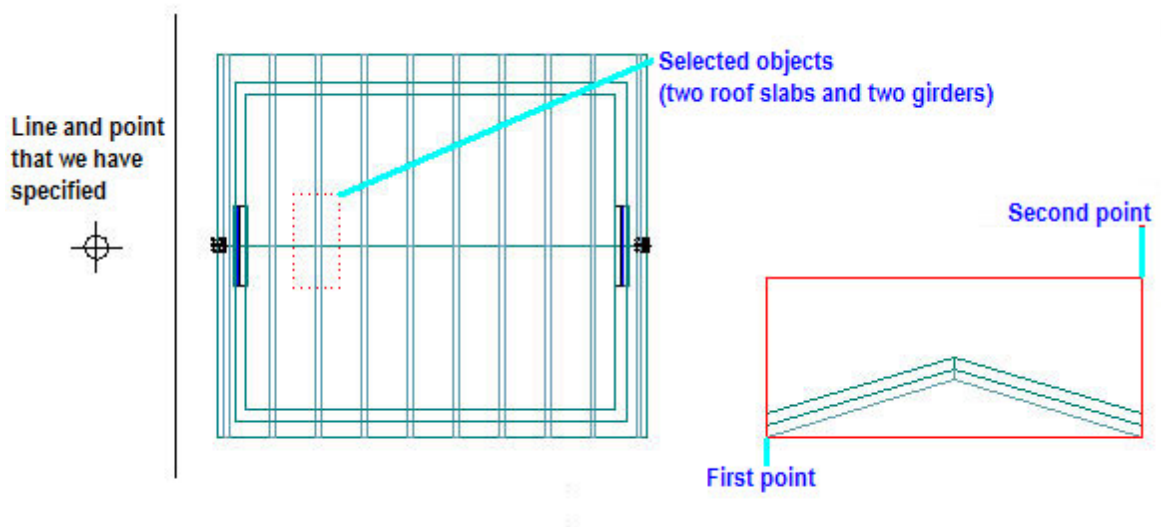
Specify the top right point of the view or Enter for real dimension:

Select objects you want generate the view:

The first question establishes the viewpoint to calculate object projection. As you can see, it offers the same possibilities as the command for generating views with the simple [GENVIEW command](#) which you should see for an explanation of the various options.

After having requested the insertion point, with the second point you are asked to indicate a rectangular area in which the object view model will be forced. You can avoid indicating a rectangle by pressing *Enter* and then generating the representation of the objects with their real size.

The last question is to select objects to be elaborated.



Note concerning [REGENVIEW command](#)

The command for updating views cannot be applied to views obtained with this command as the program cannot memorize objects which were selected during the first generation.

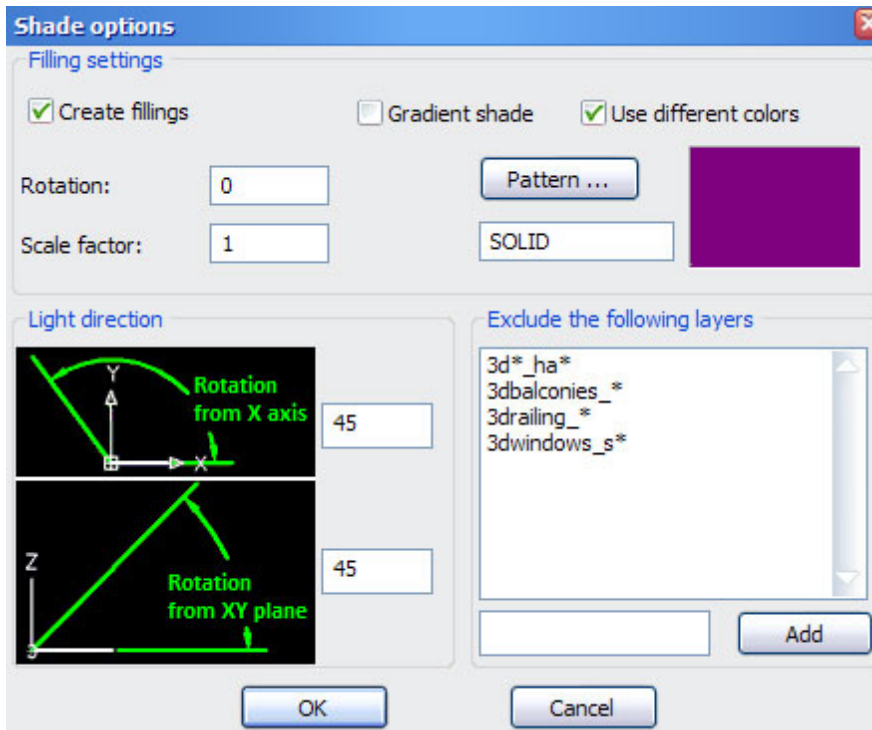
Transferred shades

Shade generation options

The generation of shades is obtained with the simultaneous generation of the 2D representation of the view.

The simulation process is similar to generation of 2D projections of 3D views obtained with the [GENVIEW command](#).

The dialog box shown on this page allows us to configure shade generation options. The window is opened with the SHADEOPTIONS command.



Filling settings

Create fillings

If this option remains disabled, only the shade contours are generated. When creating fillings, a hatch model can be chosen and in this case a rotation angle and generation scale factor associated to the model.

Gradient shade

In AutoCAD versions with which gradient filling models are generated, AddCAD can exploit these functions to generate gradient shades. This means that the color is softer where the shade ends.

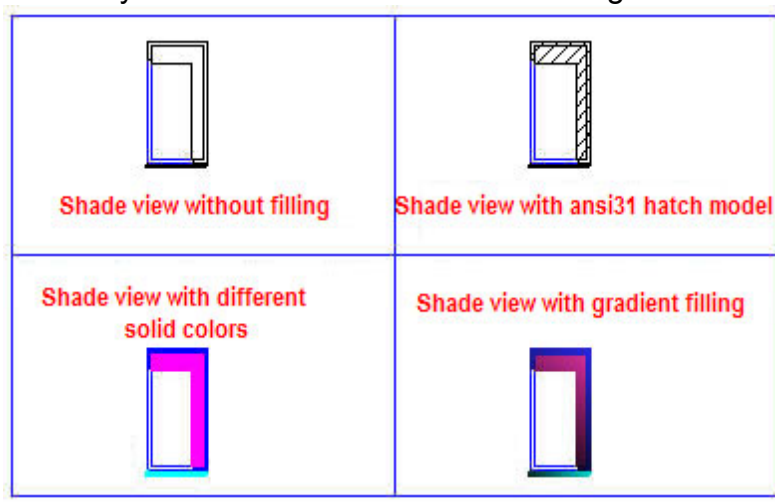
Use different colors

Each shade will have a different color depending on the depth respect to the viewpoint. This makes it easier to highlight the various details and their position in the model.

The following figure graphically illustrates the meaning of these options.

Light direction

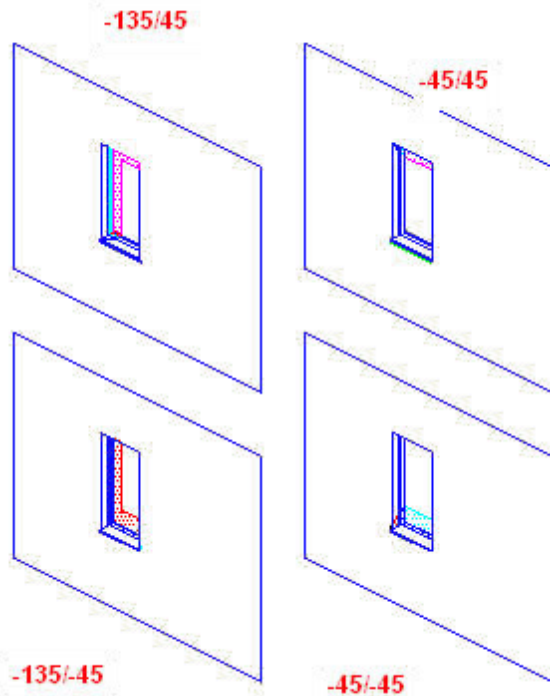
The direction the light is coming from can be set by means of two angles. In practice, the spherical coordinate system is considered with the two angles as coordinates. The first,



starting from the positive X semi-axis and rotating counterclockwise, establishes the angle which the projection of light direction forms on the XY plane. It is normally called the spherical x-axis. The second angle is the one formed by the light direction with the XY plane. This angle is also called spherical y-axis.

By combining the light direction with the viewpoint of the model we obtain the desired effect.

The following figure illustrates the model of shades on the window with the various values for the view directions.



Exclude the following layers

Processing shades is quite demanding from a calculation standpoint. One way of speeding up the process is that of excluding the surfaces of objects which are hidden from the view of what we are processing. For example, if we want shades on the external building, there is no need calculating the surfaces of the stairs inside. Furthermore, often we wish to exclude those surfaces which only create confusion on the façades. For example, calculating shadows on railing surfaces is not always a desired result. New groups of layers can be added to the list of layers to be ignored with the *Add* button. Typical wildcard can be used, such as the asterisk which stands for one or more characters. This way it is possible to quickly exclude groups of layers related to objects from shade generation. To eliminate a group from the list, just select the line and delete it by pressing *Del* on the

keyboard.

It should be noted that the surfaces of the layers excluded from the algorithm for shade generation are still processed in order to obtain the hidden lines. Consequently the 2D projection obtained will include all the entities of these surfaces.

Freezing layers

In the same way as for the view generation algorithm, even with hidden lines it is possible to freeze layers to exclude entities belonging to these layers from the generation of shades and views.

Shade generation

The generation of shades is obtained with the simultaneous generation of the 2D representation of the view. The command is similar to generation of 2D projections of 3D views obtained with the *GENVIEW* command.

The *GENSHADE* command therefore has the same options as the *GENVIEW* command. We recommend looking into the meaning of these options by studying the [GENVIEW command page](#), options which are summarized on this page.

This command is useful when, aside from the 2D representation of a given viewpoint, you also wish to generate the transferred shade model. The graphical model of the shades

obtained depends on the [options set in a specific dialog box](#). Before launching the command, make sure the set options are correct.

The command is the following.

Command: GENSHADE

First point of the section line, Enter to select an object [View/Axonometric]:

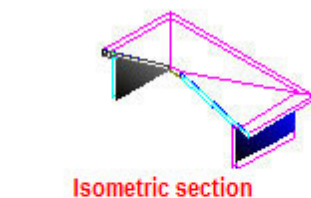
Specify the bottom left point of the elaborated view:

The *View* option elaborates the current view. If for example you have a particular axonometric view on the screen, its model can be generated with this option.

The *Axonometric* option provides you with the section axonometric view. To be able to use this option, you must have inserted the section object in the drawing. Before launching the command, you must also place yourself at the desired viewpoint in order to obtain the section from that viewpoint.

If you press *Enter*, you can select a section object which graphically indicates a section. Finally, the view to be generated can also be indicated with three points. The command requests two points where the section line should pass. Obviously if the line passes outside the building, the result will be a front view.

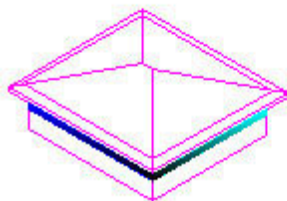
The section line alone is not enough to define the desired view. A final point is needed indicating where you wish to see the view from.



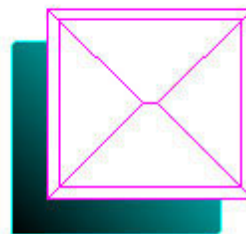
Isometric section



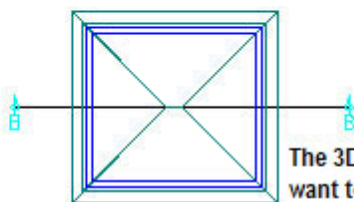
Selecting the section line object



Isometric external view



Top view



The 3D model that we want to compute

Layers used for shade entities

For lines belonging to the processed hidden view, the same GENVIEW command layers are used. The further two layers are used for the shades:

2DSHADE_L for shade contour lines
2DSHADE_H for fillings.

Advice for generating transferred shades on complex buildings

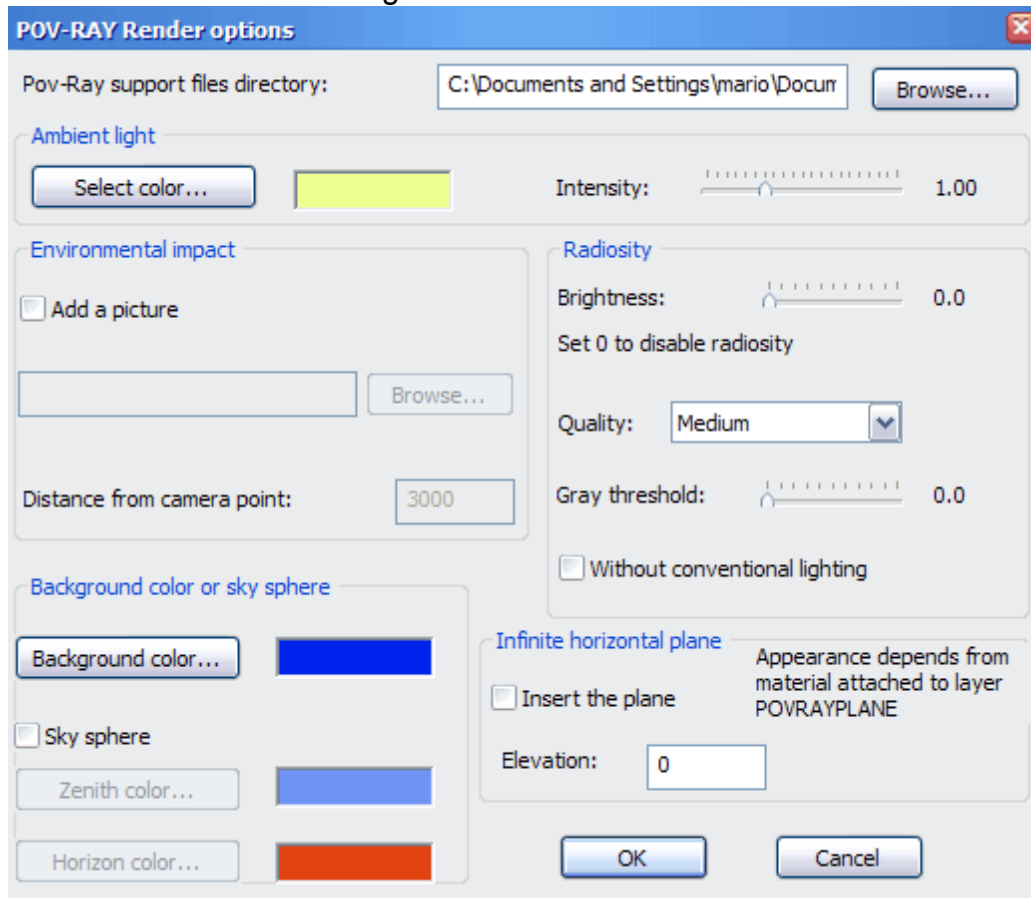
The shade generation algorithm is quite complex and may require much time to be generated for large buildings. To speed up the operation, you can save the project in a different drawing and delete the part which does not interest you. For example for front views, you can empty the building and

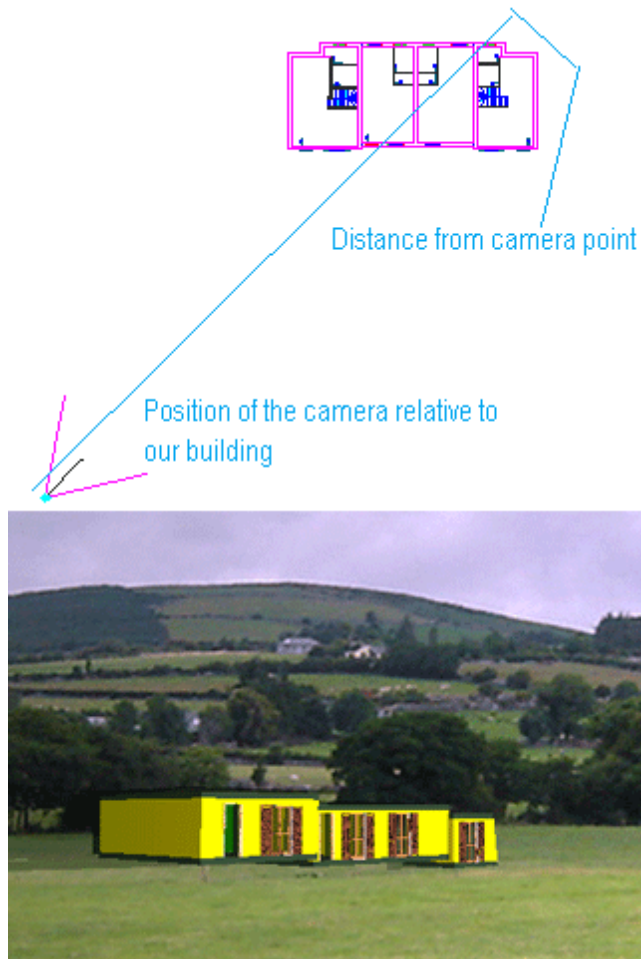
perhaps delete the back part.

Once you obtain the front view with the shades, it can be copied and pasted in the drawing where the plot layout is prepared.

Rendering options

The `POVOPTIONS` command allows you to configure a series of rendering options which make our photorealistic image closer to reality. Running this command opens a dialog box where the current settings can be seen and modified.





Pov-Ray support files directory

This is the folder containing the support files used by AddCAD to interface with POV-Ray. In this folder, for example, information regarding new materials defined by the user are memorized as well as preview images of the new materials. Generally AddCAD automatically finds the folder. We therefore recommend not modifying the folder unless it is necessary.

Ambient light

A color and intensity of a background lighting called *Ambient light* can be defined. If there is no ambient light, in POV-Ray you only see objects struck directly by light inserted in the scene. We know that reality is not like this. Optical phenomena such as light reflection and refraction also allow us to somehow see shadowed objects. The visibility of a shadowed object depends on this parameter *Ambient light* and on *Reflection Ambient light of material*, seen when defining materials. The POV-Ray manual says that the ambient view of a shadowed object is given by the result between the general ambient light value

and the ambient light effect of the object's material.

Environmental impact

A very simple and quick way of obtaining an assessment of the environmental impact of our project is with this integration with POV-Ray. You just need to take a picture on the site where the building will be constructed and insert it as a background for the project implemented with AddCAD. The formats of the files accepted by POV-Ray and namely bmp, jpg, gif and png. The position of the photo camera and its visual field respect to the theoretical position of the building are assigned by inserting a photo camera in the drawing. This is why, when exporting the model in POV-Ray, you must always select a photo camera and not just accept the current view.

The last important parameter is the distance of the image from the photo camera. The image must have a greater distance than the distance between the building and the photo camera. The value can depend on various factors and you will find the right value after a few tries (in low resolution to make it faster).



Sky sphere and infinite horizontal plane active



Just the background color

Background color or sky sphere

If you have no background image, you can use a background color or you can simulate the sky sphere with two different colors. In the first case, you need to choose one background color, in the second case two, one representing the color at horizon and the other at zenith. In both cases, colors are selected from a color selection dialog box in RGB format. For the sky sphere, the result will be a gradual transition of colors from horizon to zenith. The representation of the sky sphere can only be done if the POV-Ray model is exported by selecting a photo camera and therefore only in a perspective view. If you choose a rendering with the sky sphere, we recommend inserting the horizontal plane. Otherwise the bottom semi-sphere could also be viewed which does not correspond to reality.

Infinite horizontal plane

It is possible to insert an infinite horizontal plane in the 3D model at the *Z elevation* you choose. The appearance of this plane can be chosen by assigning

any material you wish to the *PovRayPlane* layer.

Radiosity

Rendering with the Radiosity function gives you an even more realistic result. A whole series of calculations are done concerning the light path in terms of reflection and refraction. Enabling radiosity considerably increases the time necessary for rendering. It should be done as the last stage when the scene is final. We recommend studying POV-Ray's online help in order to know all the individual parameters in detail. When using Radiosity, the parameters *Reflection Ambient light* and *Diffuse reflection* of object material have an important role.

The *Brightness* parameter adjusts the intensity of Radiosity. *Brightness* at 0 means Radiosity is deactivated. In most cases, a unit value should be ideal. Going beyond 1 could lead to unrealistic effects.

As far as quality is concerned, we have created 5 quality levels linked with appropriate values to the parameters *Recursion_limit*, *Error_bound*, *Count*, *Nearest_count*, *Minimum_reuse*, *Gray_threshold* and *Pretrace_end*.

Gray_threshold reduces color while calculating radiosity; in practice the image tends towards a grayscale. The possible values are between 0 and 1.

Without conventional lighting can be enabled when the scene has no light objects and namely the only light is ambient light. In this case the quality parameters have different values which if not appropriately modified can give way to unrealistic results.

Exporting scene in POV-Ray

There are two commands for exporting a file which can be read in POV-Ray and namely a POV file. The first allows you to directly generate the three-dimensional rendering image by automatically launching the POV-Ray program. The second simply writes a file on the

hard disk, after it has been named by the user. This file can be post processed by POV-Ray later on.

Direct POV-Ray

The POVRAYD command asks you to select a photo camera. The [photo camera](#) is the usual object to view perspectives with AddCAD. If on the other hand you wish to process the current view, respond by pressing *Enter*.

Command: POVRAYD

Select the photo camera or Enter to render the current view.

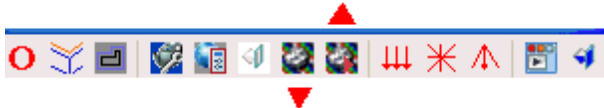
If POV-Ray is installed properly, it is launched and the image is processed immediately. The current AddCAD version launches POV-Ray correctly only if the POV-Ray version is 3.6 or higher (for example 3.6.2).

AddCAD generates a temporary file with the same name as the drawing and in the same folder as the drawing adding \$ to the end of the name. If for example you are working with a drawing named drawing1.dwg, the temporary file will be called drawing1\$.pov.

Aside from viewing the image with the set resolution, POV-Ray normally generates a BMP file with the same resolution. Again for a drawing named drawing1.dwg, we will have a file named drawing1\$.bmp in the same folder. This file can be printed or processed as you please. For example, it can be imported as an image in the drawing.

Closing POV-Ray program with this direct command

Before going back to working with AddCAD, or before you use this command again, we advise closing the POV-Ray application. Otherwise the program is launched each time you will have many POV-Rays sessions open, making it difficult to find the right one.



POV-Ray file export

The second command called POVRAY besides requesting to select the photo camera, makes a second request, the name of the file.

Command: POVRAYD

Select the photo camera or Enter to render the current view.

<choice of filename>

The POV file thus created can be processed by opening it with the POV-Ray program.

What is exported in the POV file

AddCAD considers the entire three-dimensional model made with 3dface surfaces, the entities used by AddCAD to implement 3D. In this version, AutoCAD mesh and solid objects are not supported.

Information concerning the viewpoint is inserted in the file. It can be indicated by selecting a camera or the current viewpoint can be used. Furthermore, all the scene settings configured with the POV-Ray options command are exported.

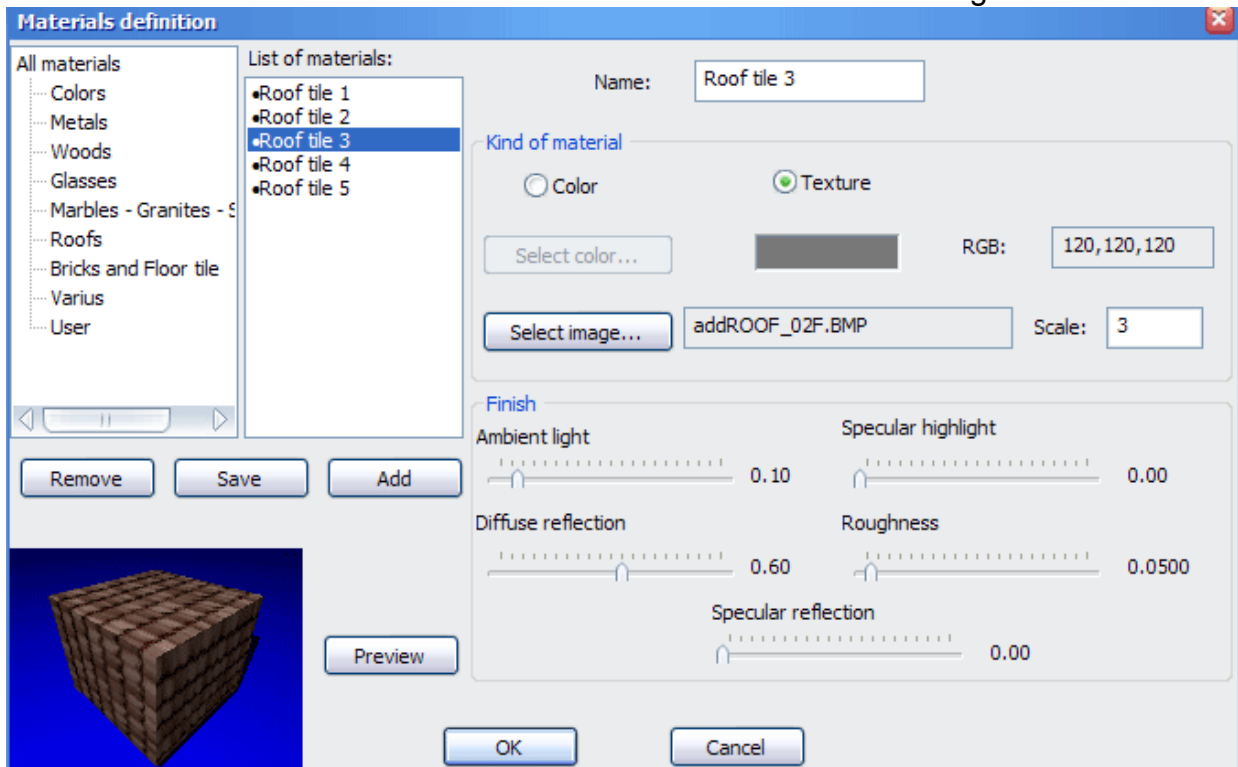
The parts of the 3D model on the frozen layers are not exported.

The lights present in the drawing are all transferred onto the file.

Definition of customized material

The DEFMATPOV command allows you to define new materials. The materials defined by the user can be either the color or based on the graphical files, bmp, jpg, gif and png. The materials library therefore contains both materials of the POV-Ray system and materials defined by the user or which we have added. The materials of the POV Ray system present in the library can neither be modified or deleted. Those which have been defined by us or by the user can be modified at any time. To make them easier to identify in the list of materials, their name starts with •.

All of the controls available to define materials can be seen in the dialog box.



The groups of materials are found at the top left. Double-click *All materials* to open the group tree. If you select a group, the list of materials of that group will be displayed. It is not possible to add other groups of materials. Once you have chosen the group, you may select a material and see its preview. If it is a customized material, you may modify the features, save them and update the preview. You will notice that the preview generated by materials defined by the user has a black background while the others have a blue background. If you write a name of material different from the one already existing, by pressing *Add* you may insert it in the list right above the selected material.



Example of texture image

Kind of material

There are two kinds of material, based on color or on texture image. If you define the material based on color, you can open the dialog box to select the color index. If you define a material based on texture image, a texture image may be chosen from the hard disk. You will notice that the selected file is used. Therefore do not cancel that file after having defined the material. It must be a good quality image but does not need excessive resolution to avoid slowing the system down. A good compromise is a resolution on 200x200 pixel and a color depth of at least 24 bits.

The Scale takes into account the ratio between the image pattern and that of the rendering result. The overall effect is achieved by combining this factor with the one assigned while assigning the material to the layers.

Transferring materials to another computer

We recommend using a specific folder where you can put all the graphical files so when you transfer materials to another computer, you only need to move this folder into the same position. You must also make a copy of the POV-Ray support folder described on the [Rendering options page](#). The file with material definitions called *addkadUser.inc* must be transferred to the new computer in the POV-Ray support file.

Finish

For each material, a series of finish parameters can be defined such as *Roughness*, *Reflection*, *Ambient light*, etc. While defining the material, a rendering preview can be generated to evaluate the appearance. The preview is created by the POV Ray rendering engine.

Ambient light

Since objects are seen only if lit up in POV-Ray, this parameter allows you to simulate diffused lighting and thereby to see the object even without direct light. Values between 0.1 and 0.2 normally provide the most realistic results.

Specular highlight

Brightness is linked to *roughness*, meaning that if a material is shiny it will be bright, otherwise it will not. Brightness is therefore a parameter which represents the capacity of reflecting light at points where light falls on the object perpendicularly. Substantially together with low roughness values, it contributes to the brightness of the material. This parameter normally has very low values, a few decimals.

Diffuse reflection

Indicates the ratio between the amount of light diffused by the object and the amount of light the object receives. A value of 0.6 means that 60% of the light received from a light source is diffused by reflection. Typical values are around 0.5.

Roughness

The roughness of the object. As mentioned when speaking of brightness, it contributes to the brightness of the object by increasing or decreasing the reflection area of light on the object. Acceptable values are generally very low, around 0.1.

Specular reflection

Represents the specular property of the object. It not only reflects light but also the surrounding objects. The value "1" represents the ideal mirror. To achieve the pure specular effect, you would need to annul the parameters *Ambient light* and *Specular*

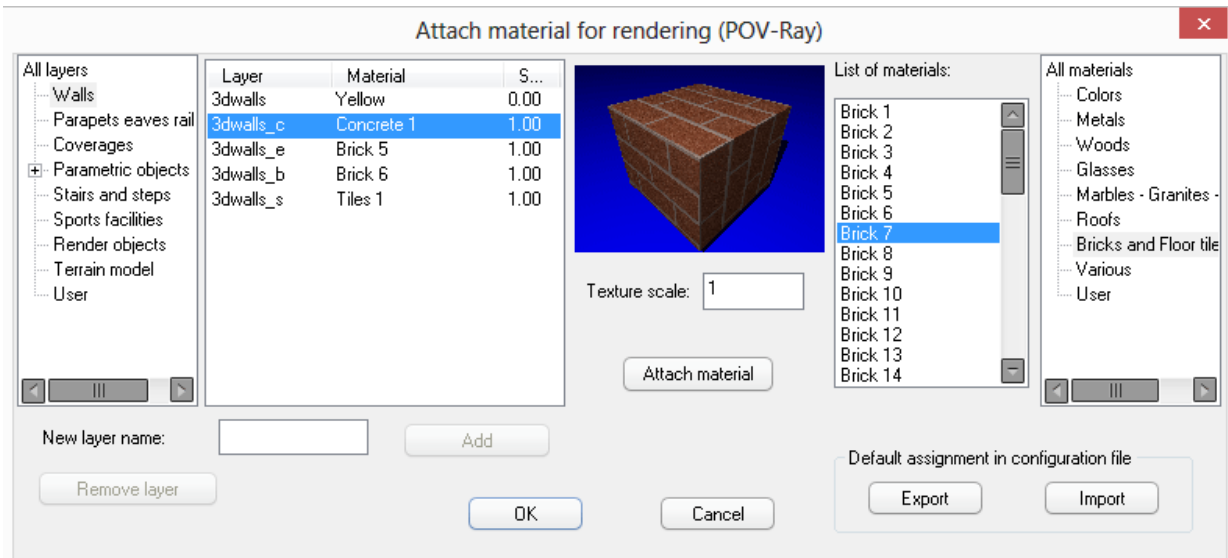
highlight. Normally a value around 0.3 already implements a glass-like specular effect. For many materials the appropriate value is 0.

Assigning materials to layers

We believe that the best way of assigning the various materials to the elements of the 3D architectural model is that of Layers. AddCAD generates the surfaces of the model by placing them on different layers. Materials can be associated to these layers. This makes it a very simple and quick method. Keep in mind that, if necessary, the [OBJECTCL command](#) changes the name of the layer by simply selecting the element of the object to which you wish to assign a different layer name. At this point, you assign any material you want to this new layer. This mechanism allows you to work freely.

POV-Ray provides you with a wide range of methods to define a material. Starting from Pigments (colors) and passing through predefined but modifiable procedures such as granite, wood, clouds etc. and arriving to textures based on graphical files. We have used many POV-Ray materials and we have also defined several of our models. The result is a complete library of materials. If you wish to know more concerning definition of materials in POV-Ray, see [Definition of customized materials](#).

Materials are assigned inside the drawing by means of a dialog box recalled by the POVMAT command.



Materials library

On the right we find the list of materials of the library. The materials are divided by categories.

AddCAD Layers

On the left is the list of AddCAD layers with the associated materials. In this window it is possible to associate the materials of the library to AddCAD layers. Just select the material on the right and the layer on the left and click *Attach material*.

Texture scale

If the material is the Color type, the scale factor does not matter. In all other cases, the meaning is evident. For example if we have material such as bricks, the size of the brick on the wall will depend on the scale factor. When a material is attached, a scale factor

congruent to that material is automatically associated to it. In order to adapt the size of the model to special requirements, it is can be changed by assigning a different value in the specific box.

For layers not present in the list on the left and which must be included in the drawing, the material used will be the layer color.

Default assignment in configuration file

Assignments are stored in the DWG file in an appropriate internal dictionary.

As you can see in the figure, however, the command allows you to change the default configuration by saving in the file the associations made in the drawing and alternatively to reassign the stored associations back in the drawing, thus recovering a standard situation. When you start a new drawing the assignments material-layer are those of the standard file.

Information for using POV-Ray

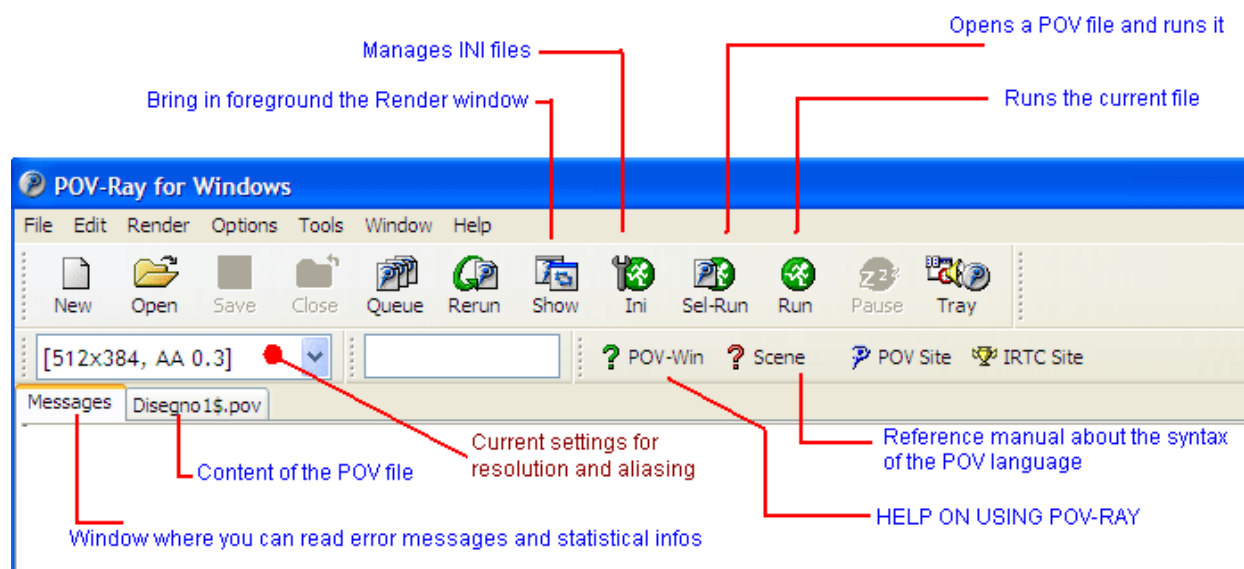
Where can I download the POV-Ray rendering engine?

<http://www.povray.org/download/>

POV-Ray has a clear and comprehensive help file. This page has useful introductory information. This information is therefore the bare essentials to achieve good results with projects made with AddCAD.

Opening and program menu

The following window has the most important parts of the POV-Ray application window.



Resolution and Anti-aliasing

POV-Ray uses the INI initialization files to manage a whole series of parameters. You may choose different configurations of the INI file directly at the top left. There are two main settings to keep in mind.

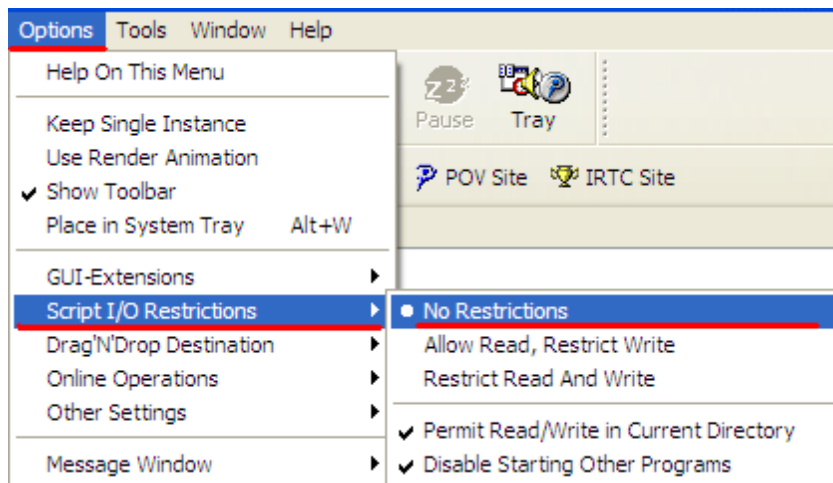
- Resolution in pixel (width x height)
- Choice of whether or not to use the antialiasing algorithm.

Higher resolution and even more so antialiasing increase processing time.

Antialiasing use a technique to reduce the 'aliasing' affect between contiguous processed pixels. The image processed with this algorithm is 'clearer' and smoother.

Image in render window and image saved on hard disk

Besides viewing the image in the render window, POV Ray saves an output file in various formats. Normally, if not specified otherwise, a 24 bit BMP file is saved (in the same folder as the POV file). This is a standard file which can be opened with all Windows applications. The BMP file will have the same resolution chosen for the rendering window.



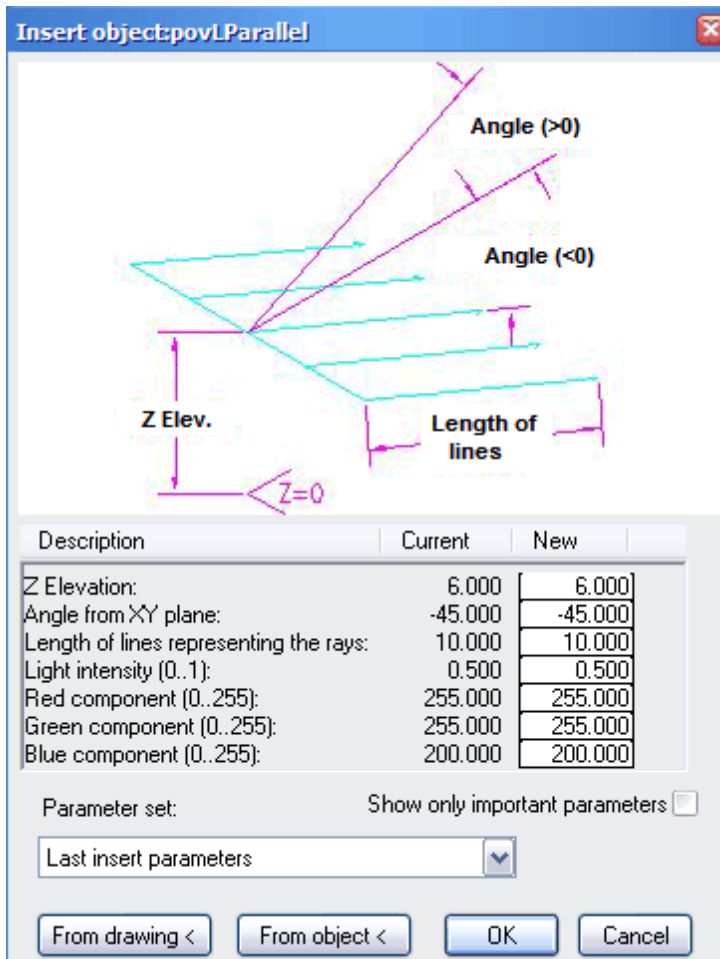
Eliminate I/O restrictions

Over time, the POV format has become a sort of script file. Therefore in the first windows operating systems which were less protected than the current ones, you could run into situations where some POV files downloaded from unreliable websites could be threats for the system. This is why POV-Ray provided this type of option. We recommend eliminating restrictions as they are no longer needed, as POV-Ray is used to process POV files written by AddCAD. The presence of this restriction causes a series of warning messages to be displayed when the POV files are opened in folders not foreseen by POV-Ray.

Inserting lights in drawing

Inserting lights in the drawing is very important both for [exporting in POV-Ray](#) and [exporting in VRML](#). In particular these objects must be used to obtain shade effects in POV-Ray.

AddCAD allows you to use three kinds of lights: parallel rays, point and spot. The physical parameters can be freely changed for each of them. From a practical standpoint, the insertion of light objects follows AddCAD's method for insertion of other parametric objects. In fact they are particular parametric objects. The type of light to be inserted can be chosen from the toolbar. It is possible to insert several types and several lights of the same kind in a scene in order to obtain a simulation as faithful to reality as possible.



Common parameters

All three kinds of lights have the intensity and light color in common. We recommend not entering values outside of the indicated intervals. If you set the intensity higher than 1, the light could be too strong. The color of the light is indicated with the RGB system. The various components have a maximum value of 255, meaning for example that a white light is (255,255,255). To conveniently choose RGB values and therefore to have a light color preview, you can use AutoCAD's color choice tool. If you go to True Color tab and choose the RGB system, you can easily copy the RGB values required for the color you want.

Parallel ray light

This kind of light is very good at simulating sunlight. A light source at an infinite distance or very far away has rays with a parallel direction. The parameterization window at the side illustrates the required data.

The *Z Elevation* and *Angle from XY*

plane values combined with the XY plane rotation angle determine the direction of the rays. It must be pointed out that the objects behind the point of insertion are also eliminated and therefore the point of insertion is only for determining the direction of the rays together with other parameters. The parameter *Length of lines representing the rays* does not affect the geometrical physical nature of the light and is only a graphical characteristic of the object in the drawing.

Point light

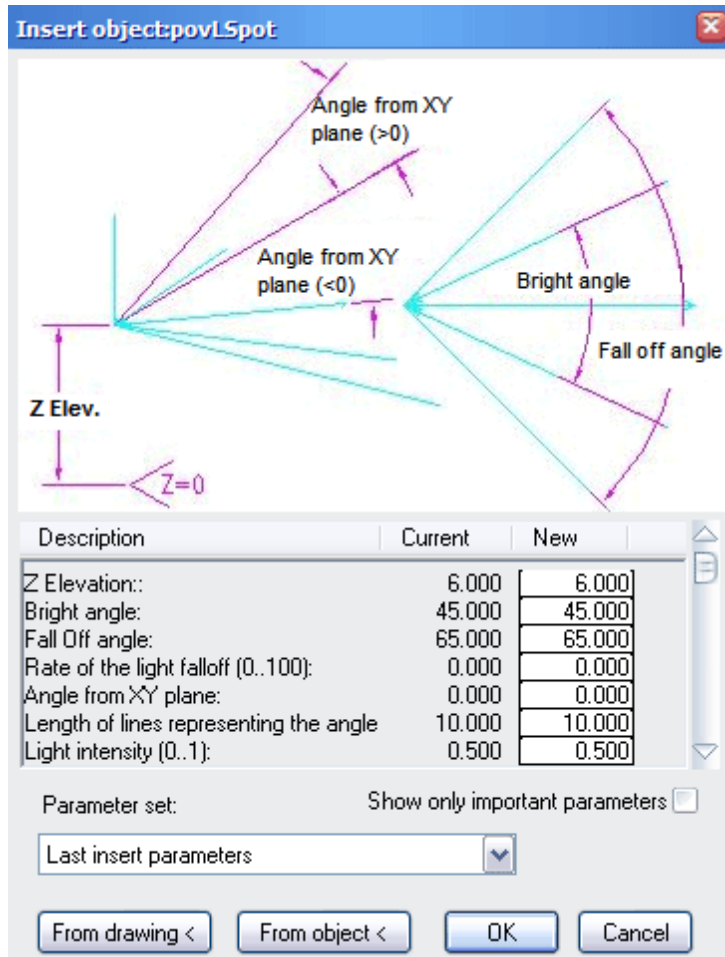
Point light is the simplest kind of light. It irradiates evenly in all directions. Therefore the point of insertion of the object is fundamental because it is the center where the light comes from. The only parameters of point light are those common to all kinds of light, namely color and intensity.

Spot light

The spot light, like the point light, has a center of radiation which coincides with the point of insertion of the object in the drawing. Unlike the point light, the spot light does not irradiate the entire space but defines a cone of light. The internal part of the cone has the maximum brightness which then reduces until it reaches zero at a certain angle of the cone. The image below shows the parameters of the object.

We need to explain the meaning of the parameter *Rate of the light falloff*. This parameter is only important for POV-Ray as a similar behavior does not exist in VRML. POV-Ray

calls it tightness. It takes into account the passage between the first internal cone and the second external cone. If the value is 0, the brightness of the internal cone remains constant throughout. Increasing the parameter causes a continuous attenuation between the cone axis and the internal angle. This makes the passage between the two zones less abrupt and 'softer'.



Exporting in VrmI format

The considerable interest surrounding this format is due to the fact that it is a standard ISO. This has allowed the implementation of so-called Plug-ins which when installed in various Internet Browsers allow the display and animation of files written in this format. The name extension of these files is wrI. The command for exporting VrmI files is called WRLOUT though its name need never be written as it is present in the Toolbars and menus.

As we said, there are several plug-ins which can be downloaded freely. Here we indicate two of the most famous links.

[Cortona3D](#)
[CosmoPlayer](#)

Il vantaggio di avere un plug-in nel browser è indubbiamente quello di poter aprire il file vrmI anche se si trova su un server collegato a Internet. Tra l'altro le dimensioni di questi

file per un disegno architettonico fatto con AddCAD consentono, con la velocità delle reti attuali, un download in tempo reale. Nessuno impedisce ovviamente di aprire il file sul nostro disco locale.

The advantage of having a plug-in on the browser is definitely that of being able to open the vrml file even if it is on a server linked to Internet. Furthermore, the dimensions of these files for an architectonic drawing made with AddCAD allow them to be downloaded in real-time considering the speed of current networks. Of course the file can be opened on our local hard disk as well.

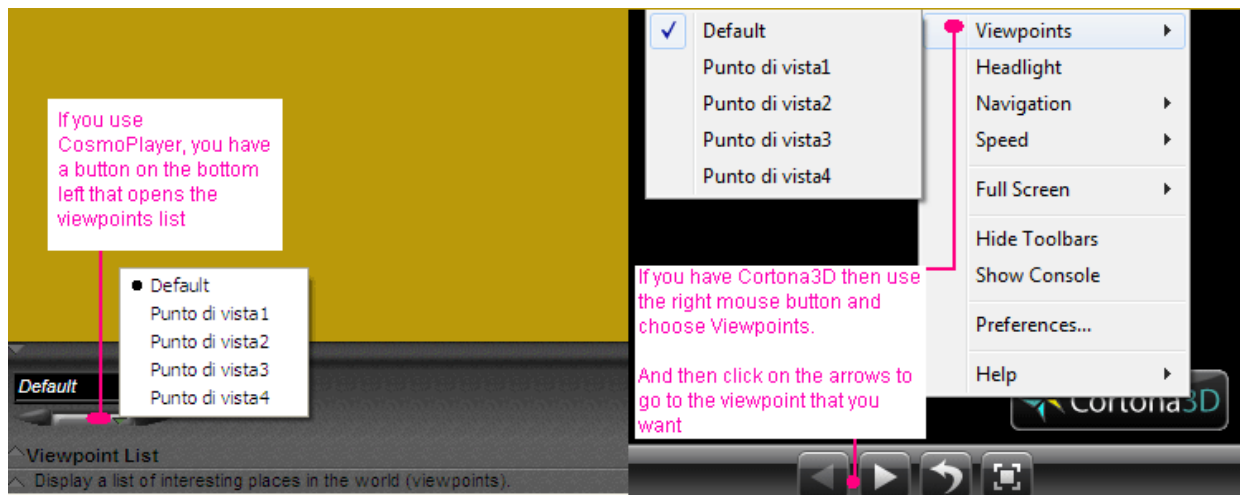
The WRLOUT command makes the following requests.

Command: WRLOUT

<acquisition of folder and filename to save wrl file>

Select the photo camera or Enter to finish:

After the file has been named, the command asks to select as many AddCAD [photo cameras](#) as you wish. These cameras generate just as many default viewpoints in vrml, which can be selected in the plug-in to begin browsing in the model from that point.



The current viewpoint inside AutoCAD is saved with the Default name.

The overall quality of the VRML environment extract depends on

- the color settings of the layer which can be modified with the [DDL COLOR](#) or LAYER command of AutoCAD
- presence of [light objects](#) with appropriate colors and intensity
- specular color, ambient intensity, shininess and transparency [settings](#).

What is exported in the VRML file

Summarizing we can say, AddCAD considers the entire three-dimensional model made with 3dface surfaces, the entities used by AddCAD to implement 3D. In this version, AutoCAD mesh and solid objects are not supported.

The parts of the 3D model on the frozen layers are not exported.

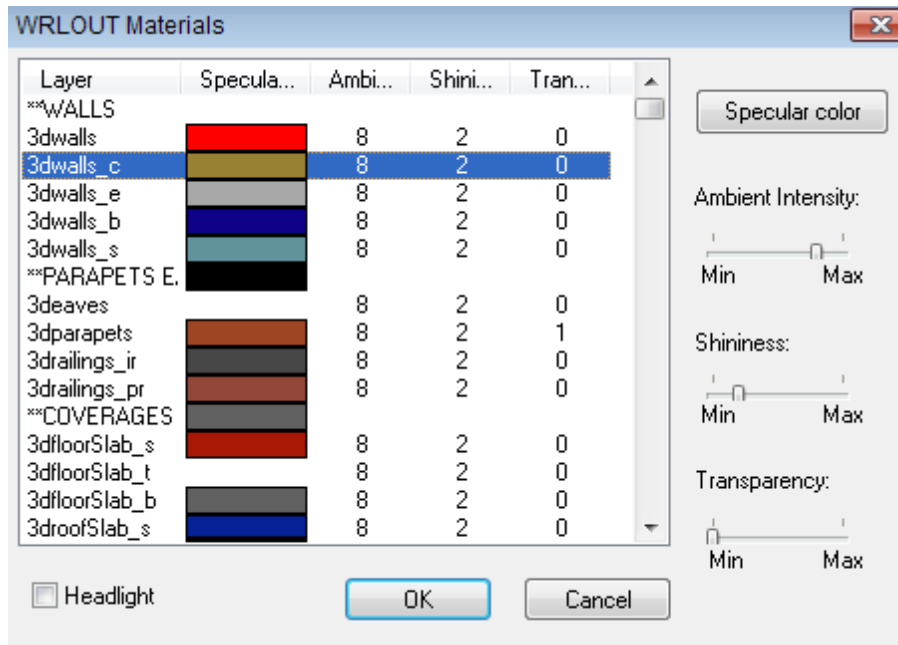
Information concerning the selected viewpoints is inserted in the file. The lights present in the drawing are all transferred onto the file and help to light up the VRML environment.

Finally the [headlight activation option](#) is set as well.

Assigning material for VRLM

This AddCAD-VRLM interface version does not support materials based on texture and images.

Using these types of materials slows down animation.



The various elements of the architectural model can be represented by using two colors and the *transparency*, *Ambient Intensity* and *Shininess* features of the material. The first is the color of the layer used in VRML for the so-called *diffuse color* and the second is a color which can be selected with the WRLMAT command dealt with on this page. The second color in VRML is called

Specular color. Calculating shades, typical of a raytracer like POV-Ray, is unthinkable for VRML animation due to the amount of calculations to be made. The combined effects of the two colors with the other features of the model elements implement a sort of play of shadows which is very interesting.

Keep in mind that the overall result of the layer color which we call the diffuse color is linked to the reflection value. This is the reflected color and therefore the color of the object. The higher the reflection, the greater the contribution of diffuse color. On the other hand, the specular color is linked to the shininess of the object and namely to the reaction to lights inserted in the drawing in the angle of incidence of the lights. The more the shininess, the greater is the weight of the specular light at the points where the incidence of the light at the surface is normal. Consequentially even the intensity of the lighting elements plays a role, as well as their color.

The diffuse color is assigned with the [DDLCCOLOR command](#) (AddCAD layer attributes configuration). The WRLMAT command allows you to assign the *specular color* associating it to the AddCAD object layers. The other ambient intensity, shininess and transparency parameters can also be associated to the AddCAD layers.

Transparency

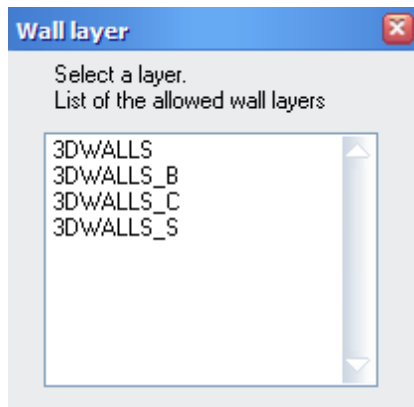
Obviously transparency is only assigned to those layers which contain transparent elements such as glass in windows and doors.

Headlight

Generally for VRML plug-ins insert a light coming from the observer's viewpoint. This light is aimed at viewing a 3D model even without lights in the model. If however we insert [real lights in the scene](#), then the headlight can only overlap with negative effects.

Changing 3D wall face layers

This is an interesting feature when you wish to attribute different colors and materials to the sides of a wall. An individual wall or a complete external perimeter can be selected with just one click. The name of the command is XLW3D. As usual the command can be activated from the toolbar, from the drop-down menu or from the multifunction bar.



Command: XLW3D

Select a line or arc of a facade or [Outside]:<Select>

.....

Select a line or arc of a facade or [Outside]:<Enter>

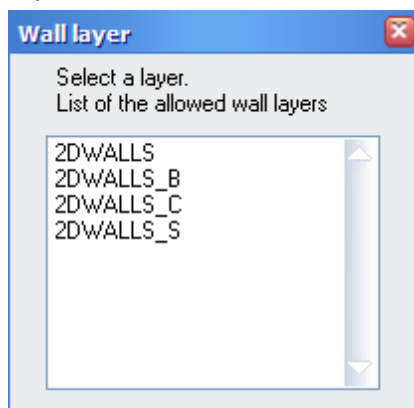
The XLW3D command performs the selection in a cyclic way.

The root of the layer name of the façade surfaces is the one selected in the list. As is known this name is joined with the prefix 'ADDC' and the floor elevation suffix.

The 2D representation must also be left visible during the selection. At the subsequent regeneration of the 3D model, the surfaces will be generated once again on the original layer. The program keeps no trace of the modification made with this command in its memory. To make layer names permanent, even in view of regeneration, we recommend using the [WALLML command](#).

Managing different layers on façades

To obtain realistic images, AddCAD allows you to associate materials to layers. As far as walls are concerned, we know that it is limiting to just one layer for the 3D wall representation. Once the wall model has been generated, there are two methods which



allow you to diversify the façade layers and thereby the materials associated to them. The [XLW3D command](#) allows us to choose the layer name to be assigned to a selected façade or to the whole outside, changing the layer of the 3D wall surfaces. The WALLML command which we will show on this page seems even more powerful. The WALLML command allows you to change the layer of the 2D plan walls (lines and arcs) on a façade or on several selected façades or on the entire external perimeter by choosing it from those available in the drawing.

This makes the assignment permanent in view of modifications to openings or walls. In fact every time the 3D wall is generated on the right 3D layer. As long as 3D Wall Layers inherited from wall 2D is checked in *Openings and walls 2D/3D Tab* of the AddCAD Options.

The command makes the following requests:

Command: WALLML

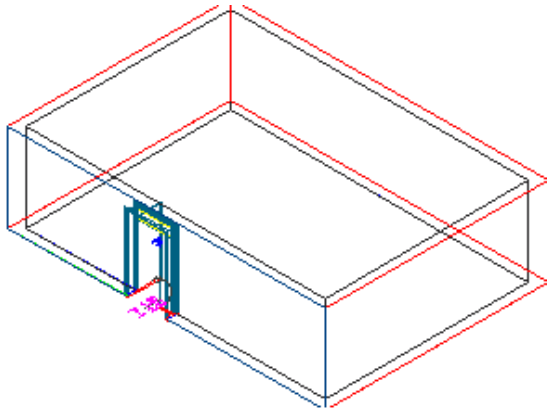
<Select the layer name from the list>

Select a line or arc of a facade or [Outside]:<Select>

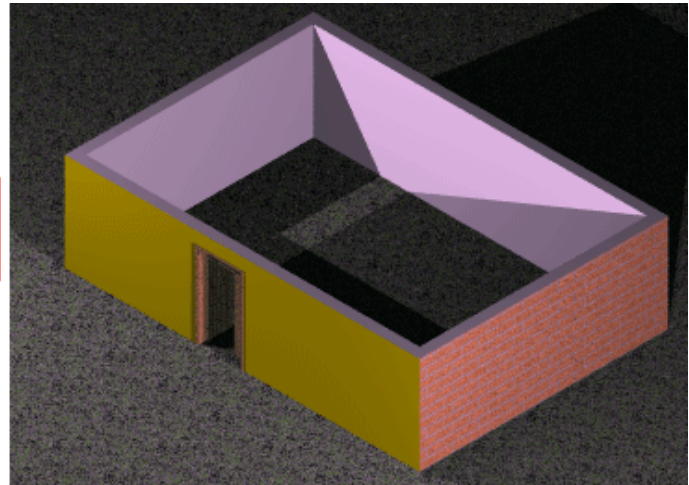
....

Select a line or arc of a facade or [Outside]:<Enter>

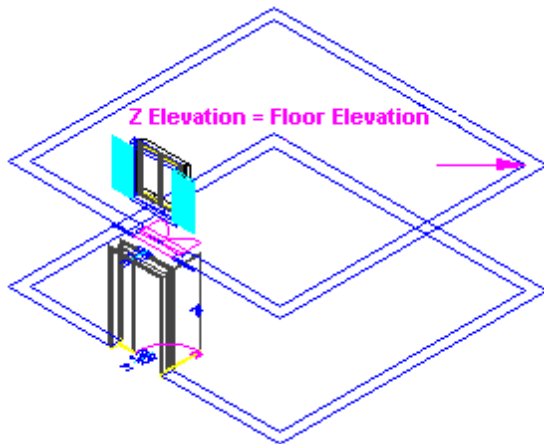
The Wallml command performs the selection in a cyclic way.



You can assign different layers to the wall face. In this way you can apply different materials to the wall face.



Floor elevation



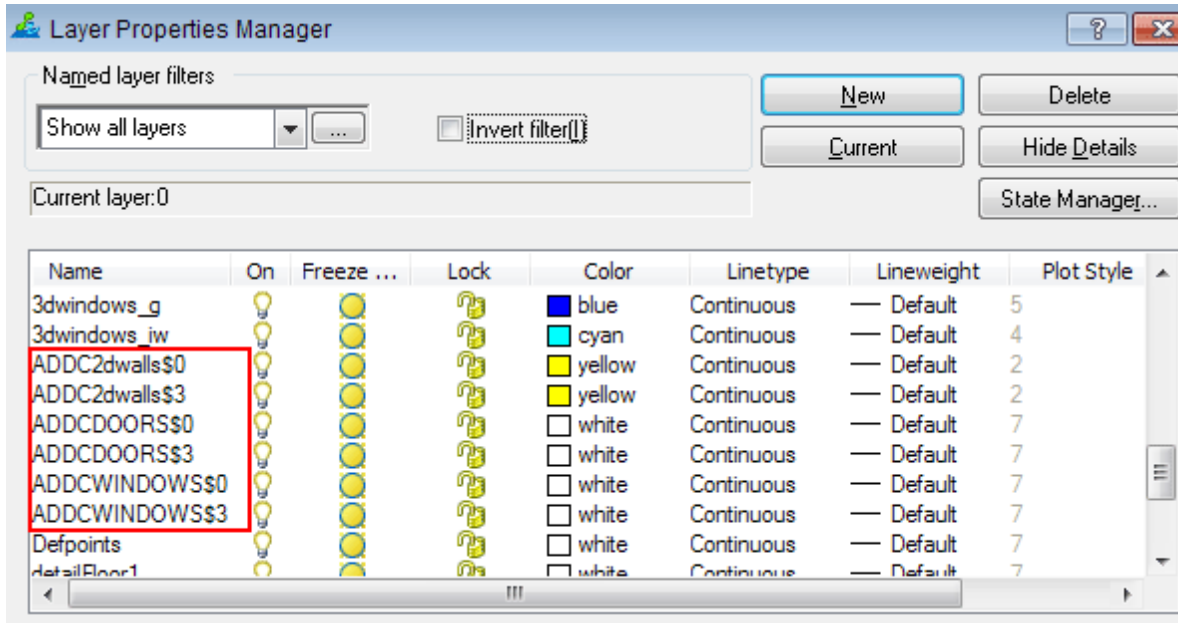
AddCAD allows you to draw by building floors. Switchover operations between floors are transparent to the user. The only data which remains in the user control is the floor elevation. In AddCAD there is a one-to-one link between the floor and the floor elevation, so they are indistinguishable from one another. The floor elevation governs everything: the dimension of the plan lines drawing, the basic insertion dimension of many groups of parametric objects; the insertion elevation of these objects ignores the current AutoCAD elevation, as the current floor elevation is still valid. This is why it is so important to know the

current floor of the drawing.

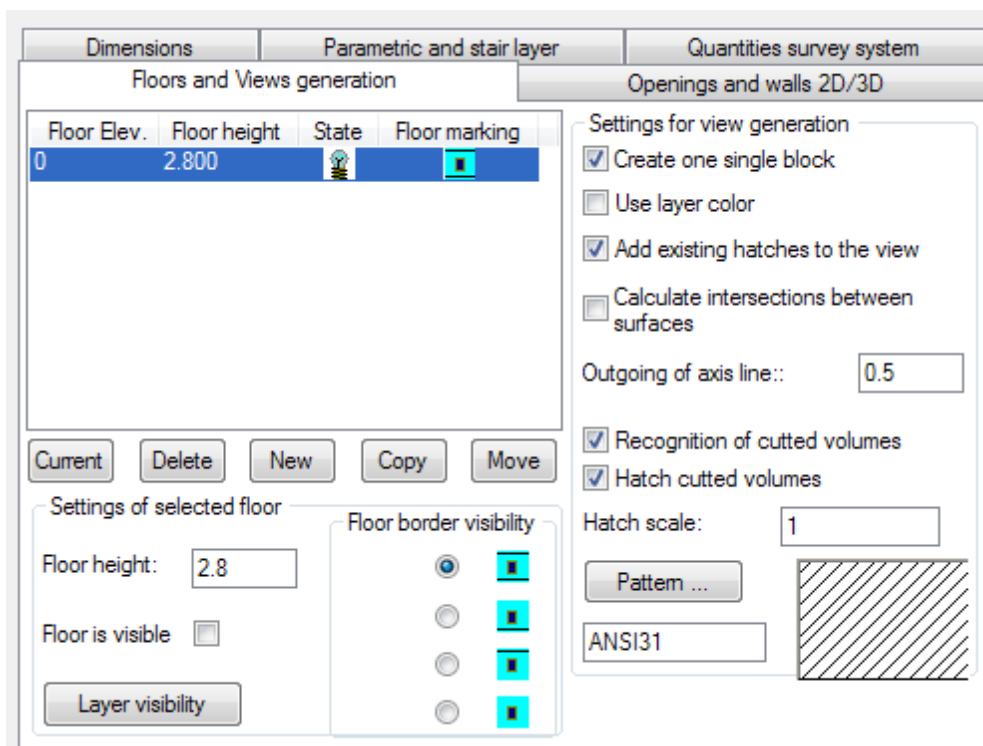
The three-dimensional development of the walls of the plan is performed referring to their floor elevation. The names of the layers created automatically (both for 2D walls, 3D walls, and the block layers of objects) have this elevation as their suffix. Some commands operate on a current floor or floor elevation; it is therefore possible to [set a current floor elevation](#) of the drawing.

When there are several floors in an AddCAD drawing, they normally overlap. There are functions which allow you to view the desired floors and to hide the others, and functions allowing you to [move and duplicate floors](#). These commands are based on a simple but smart layer name format.

Layers which contain blocks and other entities which can logically be linked to a floor have a reserved and recorded prefix called "ADDC". Their suffix is "\$" and the floor elevation.



Floors management and visibility



The choice of the current work floor, the heights of floors and their visibility are managed by *Floors and views*

generation tab of the *AddCAD Options dialog box*. Each floor is described by a row of the floors list and is identified by its floor elevation. With AddCAD, floor elevation and floor are equivalent

concepts. This is why when we talk about floor elevation and floor we actually mean the

same thing. By selecting one or more floor rows it is possible to change both the height and the 3D wall generation options and the state of visibility of the selected floors.

To make a floor current, select the line of the floor and press *Current*. The row of the current floor is always green and only one floor at a time can be current. Pressing *Delete* allows you to erase all the entities on the selected floor. It must be used with caution and its effects can be rollbacked by using the UNDO command. The *New* button creates a new floor. A floor elevation defining it is then requested. Pressing *Copy* duplicates the selected floor. The floor elevation is requested to copy the floor. The *Move* button changes the floor elevation on the selected floor moving all the objects on the floor to the new elevation. It is possible to create new floors as a copy of an existing one and to change the current floor or to simply move a floor by using the [FLOORTRANS command](#).

Changing height and moving floors to higher floor elevation

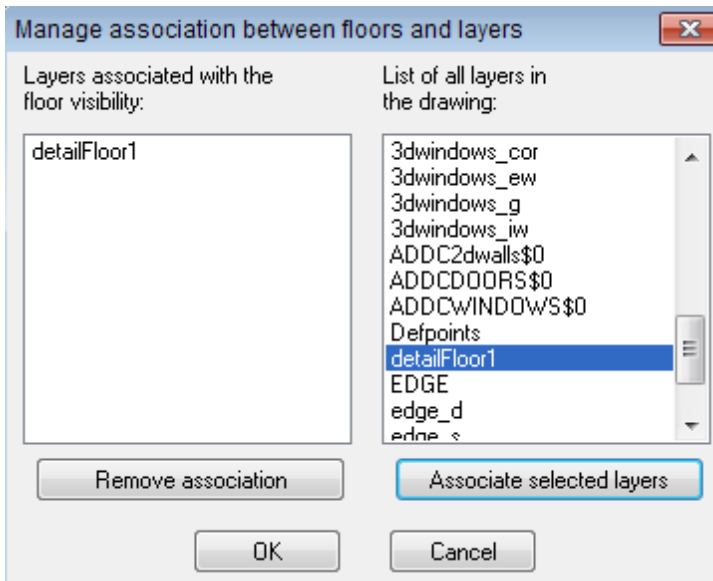
Whenever you change the height of a floor and there are floors with a floor elevation above the one whose height is being changed, the program requests if you also wish to move the overlying floors.

Floor border visibility

When a new floor is created, the 3D wall generation settings make the horizontal edges of the wall faces created visible above and below. The result is that in the front views the 'floor marking' lines are seen on all floors. You can decide how to generate the 3D representation of the walls of the various floors by using the options in this dialog box. This standard setting can obviously be changed with the [DDWALL command](#) but for individual walls which are generated in different ways, other than the standard method for the whole floor.

Dynamic updating for 3D walls

The *Apply* button is enabled if you change the *Floor height* or the *Floor border visibility*. In both cases you can update the drawing contents by simply clicking it.



Associating layers to the visibility of a floor

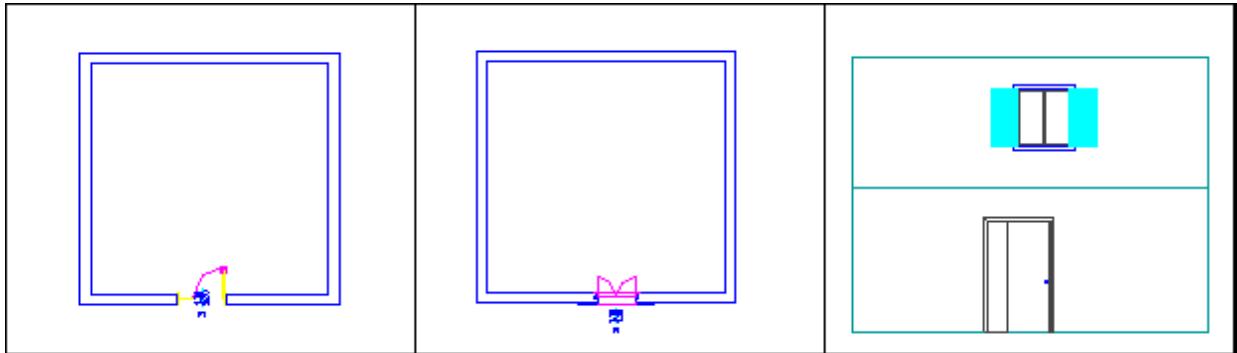
You could wish to view a certain floor with records and other details only when that floor is displayed. This function is quite useful when you prepare the plot table in the various layouts. When you select one of the floors and you press *Layer visibility*, a dialog box opens immediately allowing you to associate layers to floor visibility. The list on the right has all the *layers of the drawing*. The list on the left has the *layers associated* to the visibility of the selected floor. You can select layers in the right list

and move them to the left by pressing *Associate selected layers*. You can remove layers from the left list by pressing *Remove association*.

Model visibility and Layout visibility

It is possible to use layer visibility management in any context. If used in the Model space, obviously visibility will have effect in the Model space. If used in a *Paper space(Layout)* then the settings will be local to the Current Viewport of the Paper space. Therefore with the same method seen up to now, you can manage the visibility of floors locally in the Paper space windows. Locally means that for each window of a layout I can have a view of different floors. This function, together with the possibility of also having local [2D/3D representation](#) views of the individual windows, makes it possible to quickly prepare the layout to be plotted.

The following figure reproduces a simple plot layout example.

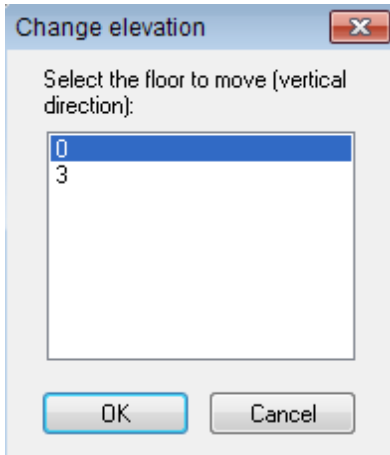


Copy and translate floors

The FLOORTRANS command allows you to translate and copy objects of a floor or an entire floor from one floor elevation to another. Another way of copying and translating floors is working in [Floors and views generation tab](#) AddCAD options. This is very important for multi-floor designs when you wish to move a floor along the Z direction and when you wish to duplicate a floor inside the same drawing. In fact very often the project of a floor of a building is similar to another floor drawn previously and often the drawing of the upper floor is obtained more quickly by modifying the copy of the lower floor. Duplicating a floor with the COPY command and translating a floor with the MOVE command is not sufficient to allow AddCAD to completely automate the process, you must also change the layer name of the objects and of the walls of the floor by modifying the floor elevation suffix. FLOORTRANS automates these operations.

Command: FLOORTRANS

Enter to traslate a whole floor or Select entities:



If the answer is *Enter*, then you wish to move a whole floor to a different floor elevation. In this case if AddCAD finds only one floor in the drawing, it requests to which floor elevation the only floor should be moved. If there are several floors in the drawing, the list of floors present in the drawing appears from which you must select the floor elevation of the floor to be moved.

Once you have selected the source floor elevation, the new floor elevation is requested which can be assigned by selecting an element of the destination floor. If the destination floor does not exist, you should write the floor elevation of the new floor.

Specify floor elevation where to move/copy the objects or Enter to select a destination floor:

Do you want delete the floor with the old elevation? <N>

Do you want change the current floor? <N>

The second question asks whether you wish to perform a pure floor translation or if you wish to duplicate the selected floor on the destination floor elevation. If you decide to duplicate the floor, then the command asks whether you wish to also change the current floor elevation. If a pure translation is performed, the floor elevation changes automatically. The program moves all the objects of the floor and assigns them to new layers the names of which differ from the old ones due to a different suffix.

In case of pure translation the old layers are eliminated automatically. To eliminate them from the drawing, you must eliminate the floor by using AddCAD's floor management or AutoCAD's LAYER or PURGE commands.

In view of what was said previously, the FLOORTRANS command is useful to start working on a new floor starting from a floor already drawn. If you want both floors in the same drawing, just use FLOORTRANS by copying the desired floor. If you want two floors, each in a different drawing, just use the following procedure.

Open the new document, copy the original floor with COPYCLIP, move it to the new document and use PASTECLIP and finally use FLOORTRANS in the new document with only translation.

Bringing objects from one floor to another

If you select objects when answering the command, they are moved or copied to a different floor elevation. This option can be used for example to begin another floor with part of an existing floor.

Multi-floor design

AddCAD is an extremely versatile tool. It does not force any drawing method. You are free to use the functionality which the program makes available in order to achieve the results they require and using the method they consider most suitable. This however does not stop us from suggesting some working practices with AddCAD. The following dilemma comes up when designing several floors.

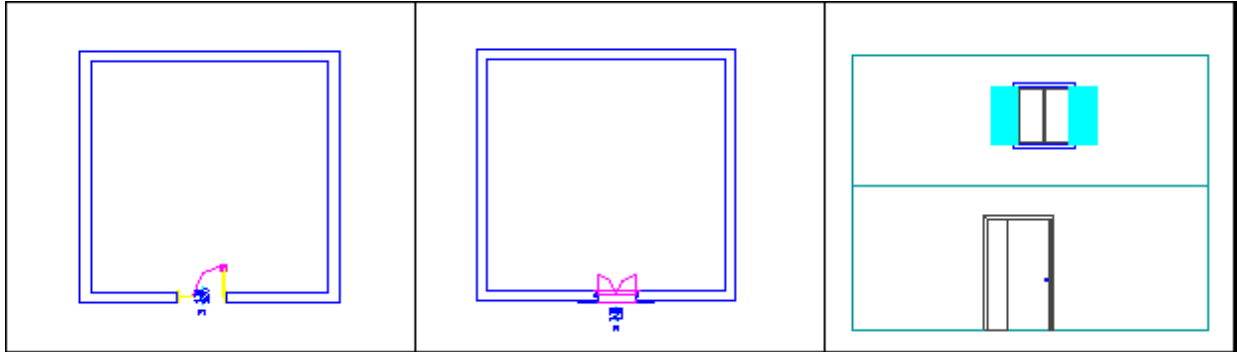
All the floors in just one drawing or one floor for each drawing and then link them all in a drawing as external references?

AddCAD allows you to use both methods. For small projects, the first method seems quicker. If you design a residential unit which is not very demanding from a graphical

standpoint, like a house, then it could be convenient to include it all in the same drawing.

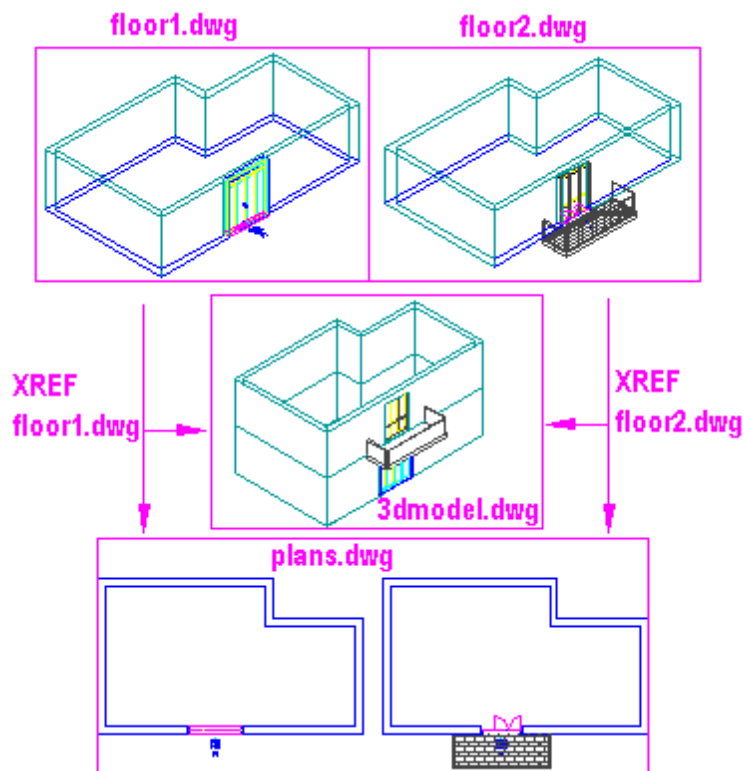
Using layouts

Using viewport in the various Layouts, it is very quick to separate the views and to plot the layout. You can obtain the separate view of the floors by [activating and deactivating the floors](#) in the individual viewport. The separate view of the 2D and 3D representations is obtained by using the [VXD command](#) in the various windows.

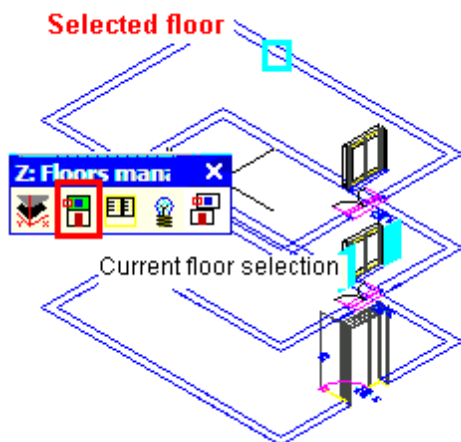


Using external references (xref)

For more complex situations, we propose the second method. Drawing all the floors of a building in just one drawing (dwg file) for challenging projects can be difficult to manage, both due to the volume of the entities to be regenerated and management of the numerous floors. Actually, especially during the design phase, it is better to work with a drawing containing just a few floors. Visualizing and assessing the overall 3D model, as well as copying the views of sections and front views and performing rendering and plots of the 3D model are operations which can be done in a new drawing, in which all the drawings of the individual floors are linked by means of AutoCAD's xref command. The plot layout can be prepared in another empty drawing, again with xref inserting the various floors moved directly to their positions. This strategy is shown in the figure. To make it easy to explain, we will deal with a simple two floor example.



Current floor



The command allows you to make the selected floor the current floor. You are requested to select an object belonging to the floor to be rendered current.

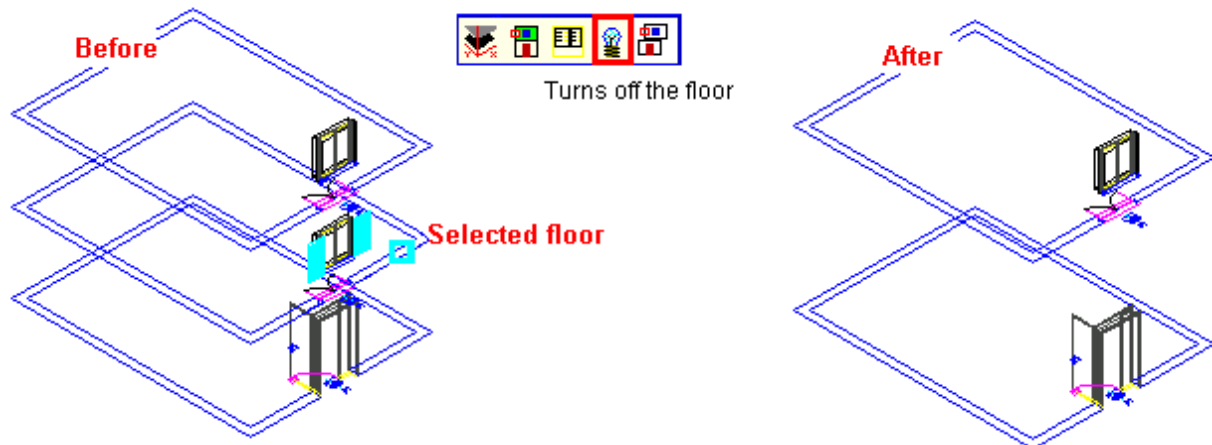
As is known, the current floor is an important item for the insertion of floor objects and the wall drawing.

Turn off floors

The command allows you to turn the selected floors off. You are requested to select an object belonging to the floor to be turned off.

Command: FLOOROFF

Select the object that belongs to a floor to turn off:



Select a floor

With this command you can select all the [objects belonging to a floor](#). A whole floor is selected by selecting any object belonging to the floor. For example you can select a wall line or a window. Once a floor has been selected, we can move, copy or create a selection group which can be used with all the commands by simply answering the selection requests with Previous.

Generally, it is not recommended to move or copy a floor. The 3D model would not be generated in the right positions. Nonetheless users have often shown interest in managing the plans of the individual floors more freely to meet their needs.

The command makes the following requests.

Command: FLOORSEL

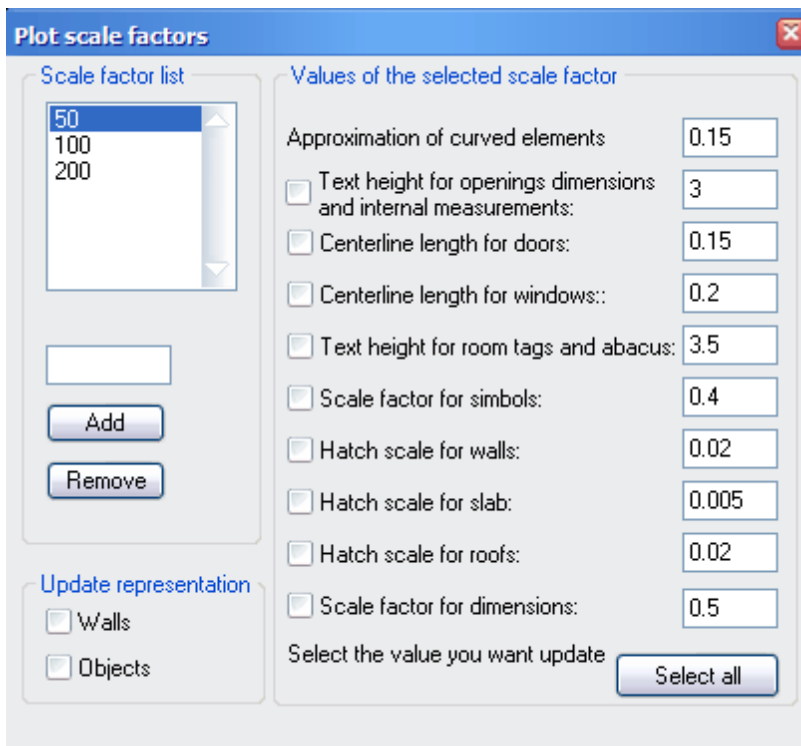
Select an object of that floor that you want specify:

Press Enter for just creating a selection set or [Move/Copy]:

By means of the *Move* and *Copy* options, it is possible to move or copy directly. Whereas by pressing *Enter*, a selection group is created which can be used later on with the *Previous* option during selection.

Plot scale factors

The SCALEF command controls in a centralized way the dimensional values of the height of some texts, the centerlines of openings, of symbols, of hatches and of dimensions. It is possible to choose the current scale factor and to update the drawing based on the desired plot scale factor.



Scale factor list, Add and Remove

It is possible to define a new scale factor. The name of the scale factor must be a number and it must represent the plot scale factor. The ten values in the right-hand column are associated to each scale factor. By selecting a scale factor in the list and then by pressing *Remove* you may eliminate a scale factor from the list. By writing a new number in the edit box and pressing *Add* you may add a new scale factor to the list. By modifying the values to the right, it is possible to modify the values associated to the selected scale factor.

Values which can be associated to a scale factor

Approximation of curved elements represents the size of the flat elements which approximate the generation of curved elements in the drawing: slabs, walls, curved openings, etc.

Text height for openings dimensions and internal measurements expressed in millimeters is intended on the plot and refers to the internal measurements of the rooms, and to the parametric opening, window and door dimensions of AddCAD.

Text height for room tags and abacus, intended as the text height on the plot, refers to the texts of the tags, room tabs and frame tabs.

The Centerline length is expressed in the graphical units of the drawing.

The Scale factor for symbols regards the electric and hydraulic symbols and in general all the blocks inserted in the form of symbols.

The Hatch scale for walls is the one implemented with the HWALL command; the other two hatches are implemented by the commands managing the roofs. The scale factor required for a correct plot can depend on the hatch model used.

The Scale factor for dimensions is the one used by AddCAD's dimension commands.

Except for the height of the texts, the factors have no direct links with the size of the paper. Personal preferences and some attempts allow you to adjust these values correctly.

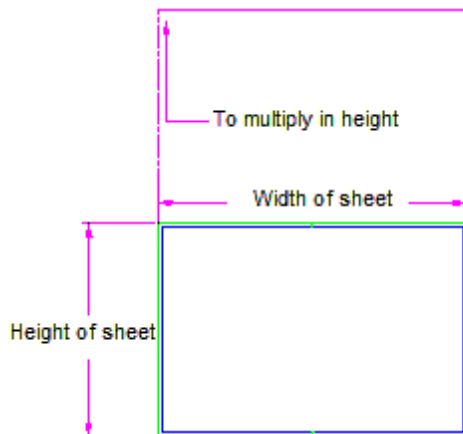
Regenerating models

After having exited by pressing OK, the behavior of the command depends on the position of the various control boxes on the left side of the descriptions. They are used to choose whether or not to regenerate objects in the design depending on those variables. If they are all disabled, then the new scale factor will only be applied to new elements created in the drawing; namely the entities already in the drawing will not be modified. If on the other hand the control boxes are enabled, then all the entities depending on the dimensions of the enabled buttons will be regenerated.

Walls and Objects Representations

It is possible to regenerate [wall stratifications](#) and the [representation of objects](#). All you need to do is exit with the relative box is enabled.

Sheet squaring



Squaring

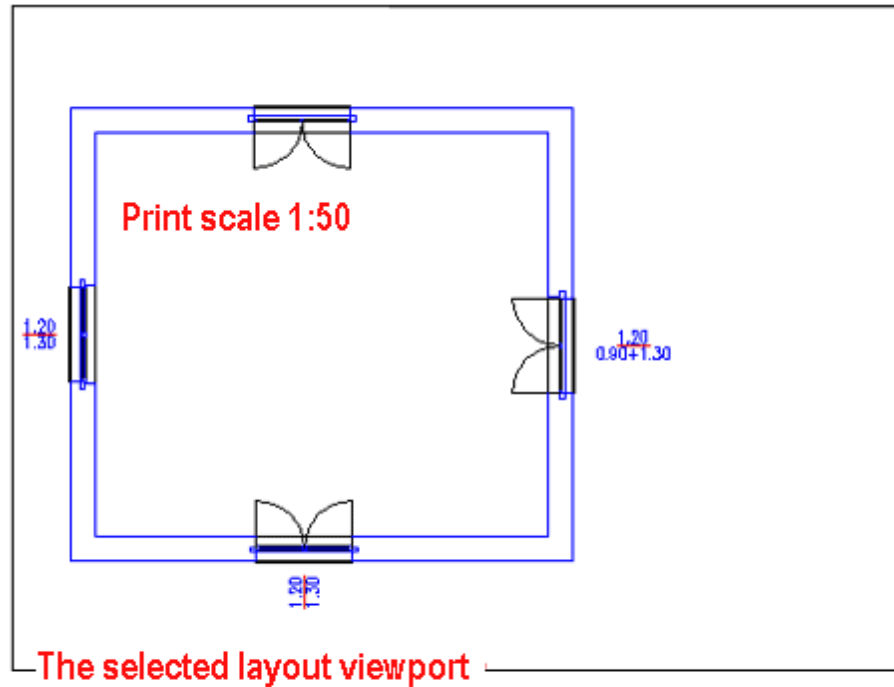
AddCAD allows you to insert parametric objects in the drawing to square the plot sheet. The figure illustrates the required parameters. The desired sheet can be selected by choosing the type of object. You are also requested the scale factor with which you would like to plot. The object has 29.7 x 21 cm column dashes. Remember that the object can be modified at any time using the [MODIFY command](#).

It can be inserted from the drop-down menu *AddCAD Draw>Sheet squaring*. There are

two models depending on whether the squaring must be inserted in the model space or in the paper space.

Scale factor for layout viewport

The habit of arranging the [drawing plots in Layouts](#) rather than in the model space plot is ever more widespread. A piece of paper and plotting options can be associated to the Layout. You can open as many windows as you like on this sheet of paper. Each window represents an area on the sheet where the desired project you can be plotted. The VPSCALE command allows you to quickly assign the plot scale factor to the selected windows by directly indicating the desired scale factor. Obviously if you zoom inside the window, you lose the scale factor and the command must be used once again to restore it.

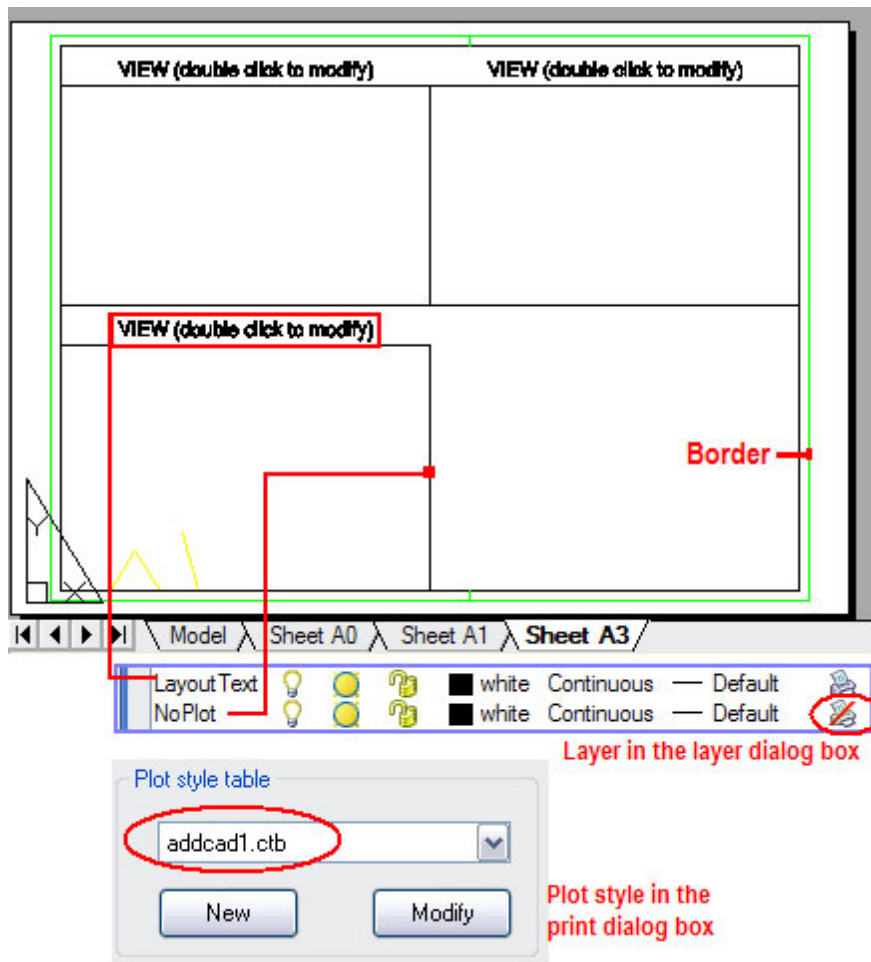


```
Command: VPSCALE
Plot scale factor:50 Scale factor
Select the window to which apply the plot scale factor:
```

AddCAD drawing templates

Default layouts

The layouts in the AddCAD drawing templates have a setting which guides and speeds up the plotting phases. As usual, you may customize formats, colors and plotting styles as you wish. However the drawing templates supplied with AddCAD represent a starting point as close as possible to your requirements.



Layout bar

There are already three layouts in the layout bar with the most common sheet formats, namely A0, A1 and A3.

Squaring

Each sheet has squaring. This is the [Border parametric object](#) which can be modified with the Modify command.

Layout windows

These are the elements which are modified bit by bit according to the dimensions of the project and of the plot scale factor. These objects which allow you to have different views on one plot sheet are common AutoCAD objects which you usually do not want to print. This is why we have defined a layer to which these windows are assigned. The layer is called 'noplots' and is not plotted.

Texts in paper space

These texts usually indicate the view plotted in the window below. They can be modified by double-clicking them and they are placed on the LayoutTexts layer.

Plot styles

The Addcad1.ctb plot style is associated to each layout. It is possible to change plot style but in that case we recommend also changing the color configuration of the layers to correctly transform thicknesses.

Plot styles

The colors of the layers can be managed with the [DDL COLOR command](#). This command memorizes the colors, thicknesses, etc. settings in a file named `layerc.def`. This file is always present in the AddCAD program folder. Obviously if a color configuration is provided, a thickness can be defined for each color and therefore a plot style defined per color. AddCAD provides two plot styles, one of which is set by default in the model drawings. This style exploits the color settings of the `layerc.def` file installed with the program.

The other style present in the styles list is `Addcad2.ctb`. The following is the thickness assignment tab of the two plot styles. The colors of the tab are all plotted black with relative thickness. The subsequent colors are drawn with the color and thickness of the graphical entity.

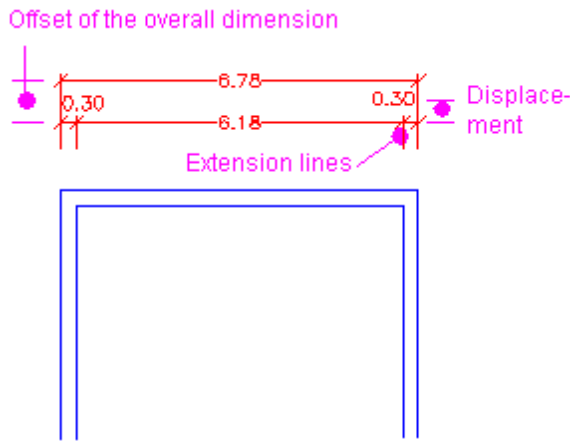
Color number	Color	Line thickness AddCAD1	Line thickness AddCAD2
1	Red	0,35 mm	0,40 mm
2	Yellow	0,60 mm	0,40 mm
3	Green	0,25 mm	0,30 mm
4	Cyan	0,40 mm	0,30 mm
5	Blue	0,25 mm	0,25 mm
6	Magenta	0,40 mm	0,25 mm
7	White/Black	0,35 mm	0,18 mm
8	Dark gray	0,15 mm	0,18 mm
9	Light gray	0,15 mm	0,15 mm
10		1,00 mm	0,15 mm

Introduction

The dimension commands allow you to dimension the project views. There are two commands to automatically dimension the inside of the rooms. The first is based on AutoCAD dimensions, while the other exploits parametric objects. The plan drawing can be dimensioned with the [DPLAN command](#) and the elevations with the [DELEV command](#). The DPLAN command has options which allow you to dimension other views. Then there is the [DV command](#) which allows you to dimension by vertices. Many commands allow you to add the total dimension alongside the automatically generated dimension chain. The prototype drawings supplied contain some dimension styles that can be used by the dimension commands.

For the DROOM and DPROOM commands and the DPLAN command face option to work correctly, you must make sure that the floor elevation value is correct for the floor you wish to dimension.

Dimension options



Some parameters for the dimension drawing are configured in the *Dimensions Tab* of AddCAD options.

Minimum value

The Minimum value is the value below which dimensions are not generated. If zero is placed, everything is dimensioned.

Length of extension lines

Indicates the length of the dimension extension line.

Move if less than

Move if less than indicates the value below which dimensions are moved out of the dimension line and *Displacement* indicates

the movement entity.

Overall dimension offset

When you choose to add the overall dimension as well, Overall dimension offset indicates the distance between the two dimension lines.

Dimensions

Minimum value:

Length of extension lines

Move if less than:

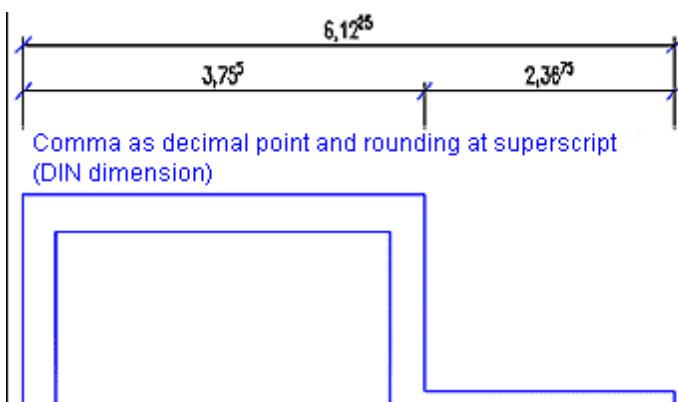
Displacement:

Overall dimension offset:

☐ DIN dimension

☐ Comma as decimal separator

☐ Round to 2,5mm



superscript of the measurement.

DIN dimension

In some countries where AddCAD is used, it is necessary to dimension according to DIN regulations. This is why we have introduced the possibility of dimensioning according to DIN regulations. This means being able to use the *Comma as decimal separator* and *Round to 2,5mm*. This fraction must be written in the dimension text at the

Room dimensions

Room label dimensions

Distance from walls:

Number of decimals:

☐ Ignore functional lines

Openings

☐ Take into account openings

☒ Whole walls

Label style

Four preview boxes showing different dimension line styles with the word 'Dimension' in pink. The second box from the left is selected with a blue circle below it.

OK Cancel

the distances between the various openings on the same wall.

The DROOM command allows you to dimension the inside of rooms. The command draws a dimension line and a dimension text parallel at a certain distance from the walls. It is not possible to dimension the curved wall of a room with this command. It is possible to establish at which *Distance from walls* the symbol and dimension text should be drawn.

Openings

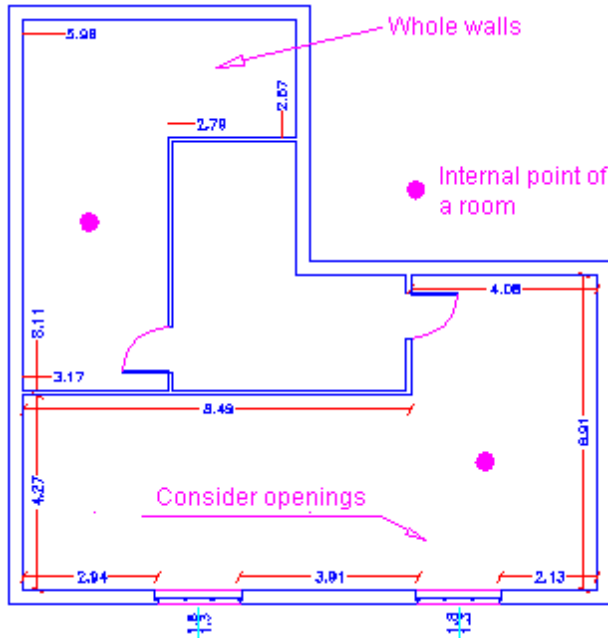
It is possible to choose whether to dimension only the length of the wall or also the openings on the walls. In fact you can dimension

Label style

The dialog box has four images representing the kind of symbol and dimension. You can select the desired symbol by pressing the button. You can also set the number of decimal places.

Drawing units/dimension units ratio

It is possible to draw in centimeters and to dimension in meters or vice versa. To change the drawing unit/dimension unit ratio, you must modify the DIMLFAQ variable of AutoCAD which already takes care of this task in AutoCAD dimensions. The DIMLFAQ variable belongs to the current dimension style.



Dimension text height

The size of the text depends on the value specified with the plot scale factors [SCALEF command](#).

Once the window is closed by pressing OK, just touch a point inside the room to be dimensioned and the command will continue automatically. Lastly you are requested to select dimensions considered redundant or which can create confusion in the drawing. They must be erased.

Command: DROOM

Pick internal point of a room:

Select dimension symbols to delete:

It must be noticed that if the room to be dimensioned is not well delimited by walls and openings, the command will stop and an error message will be triggered.

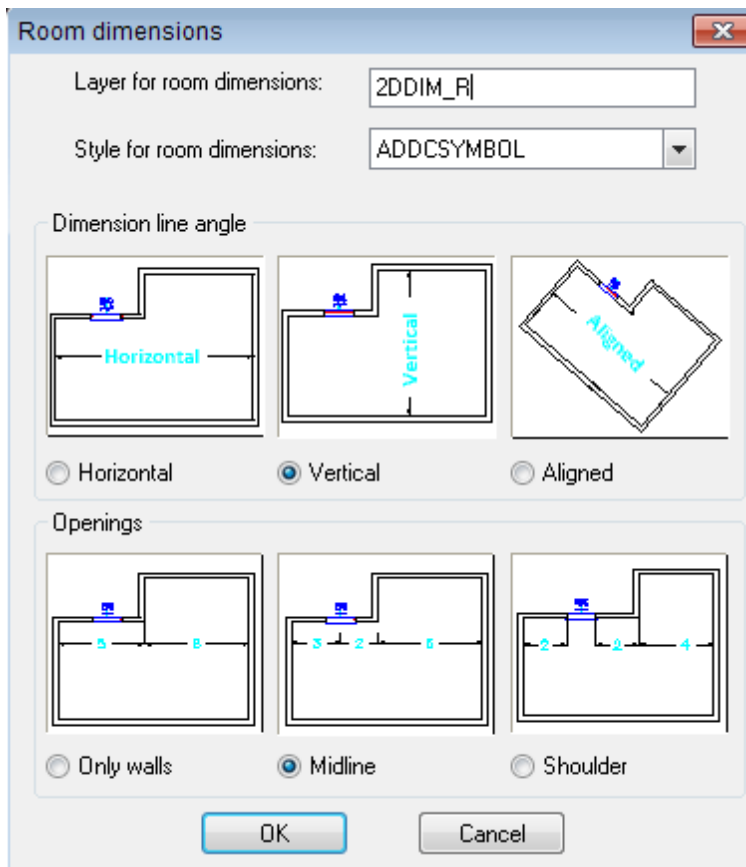
When the shapes and dimensions of rooms are modified, the command can be reused directly to dimension the inside without first needing to delete the dimensions created previously. This means that after having indicated the point inside the room, the command cancels any dimensions found inside of it.

The dimensions generated by this command are blocks generated starting from definitions of parametric objects.

Layers used

The lines and text belong to two different layers. The lines go on the 2DDIMENSIONS_L layer, while the texts go on the 2DDIMENSIONS_T layer. The dimension block set goes on the layer with the name composed of ADDCINTERNALDIM\$ plus the floor elevation value.

Internal wall dimensions



The DPROOM command represents an alternative to the [DROOM command](#) to dimension the inside of rooms. This command is based on the AutoCAD dimensions system. The dialog box shows how to choose the dimension line angle and how to dimension any openings which should face inside. If you choose *Horizontal* or *Vertical*, the program will request the point where the dimension line must pass; if you choose *Aligned*, prior to the point request, you will be requested the alignment angle.

Text and arrowsymbols dimensions

The size of the text and of the arrow symbols depends on the dimension style and plot [scale factor settings](#).

Layer for room dimensions

The root of the name of the layer used to insert created entities is

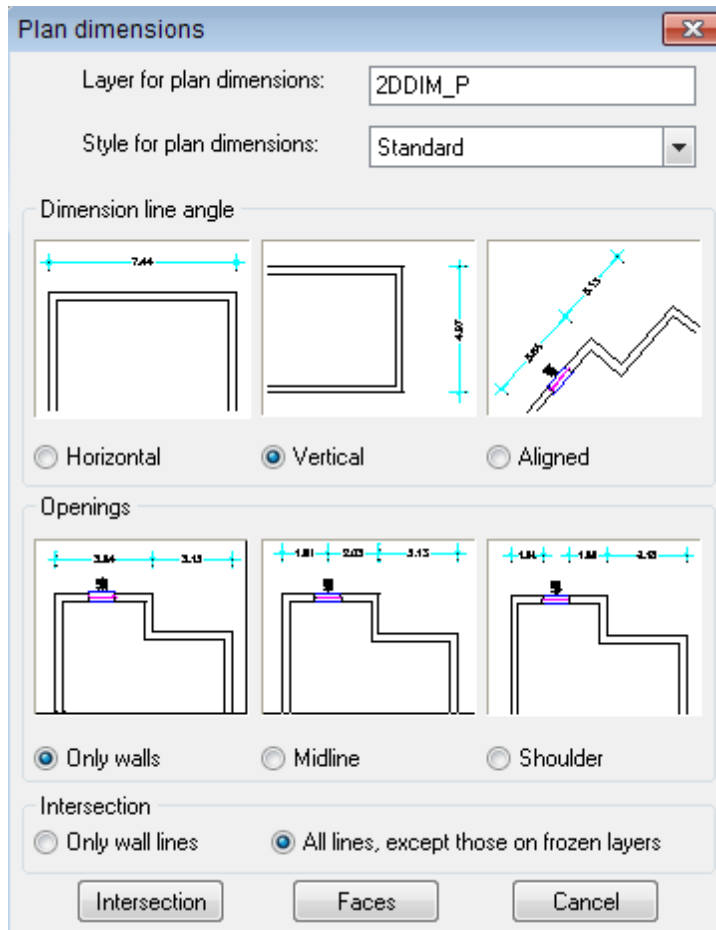
indicated this field. Like all other AddCAD objects linked to the floor, the root is joined with the prefix ADDC and the suffix is the same as the floor elevation.

Style for room dimensions

It is possible to choose the dimension style directly from the dialog box.

As far as the point request is concerned, you must pay the utmost attention and select it as close as possible to the wall you wish to dimension. Last of all if the result is a chain of dimensions, you are asked if you wish to draw the overall dimension as well.

Dimension of plans



The DPLAN command generates AutoCAD dimensions. Some parameters which affect the dimension positions can be configured in the [Dimensions tab](#) of AddCAD options.

Text and arrow symbols dimensions

The size of the text and of the arrow symbols depends on the dimension style and [plot scale factor settings](#).

Style for plan dimensions

It is possible to choose the dimension style directly from the dialog box.

Layer for plan dimensions

The root of the name of the layer used to insert created entities is indicated this field. Like all other AddCAD objects linked to the floor, the root is joined with the prefix ADDC and the suffix is the same as the floor elevation.

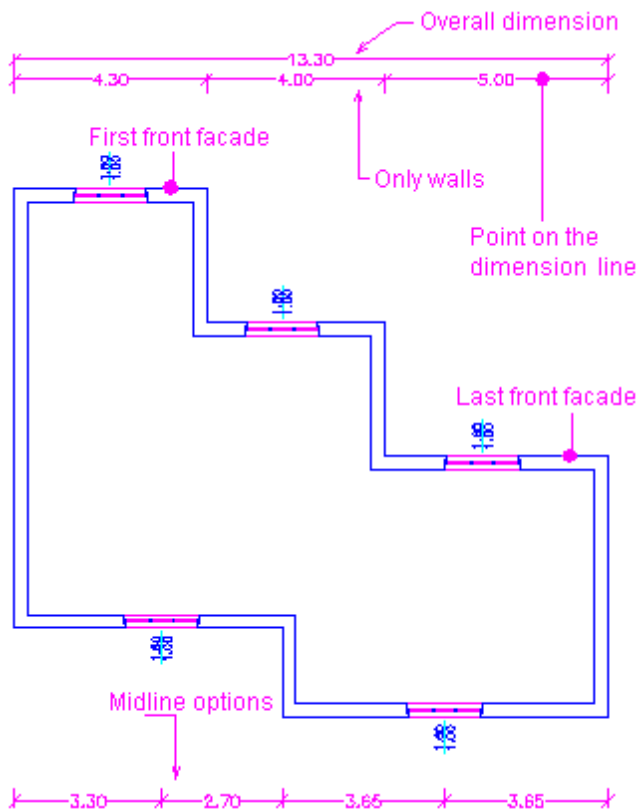
Dimension line angle

It is possible to establish the dimension line angle. The option *Aligned* allows you to dimension

while freely directing the dimension line.

For plan dimensions, you can exit by pressing *Faces*; in this case you can decide what the command must do if there are openings. In particular, you may choose whether to dimension the position of the openings, and if so whether to dimension the *midline* or *shoulders* of the openings.

The *Faces* button allows you to automatically dimension a series of continuous faces looking out on the same front view. As far as dimensioning the openings is concerned, the command works according to what is specified in *Openings* of the dialog box. The following is part of the dialog for the *Faces* method.



Command: DPLAN

Specify a point on the dimension line:

Select the first line of the front facade:

Select the last line of the front facade:

Do you want the overall dimension? (Y/N) <N>:

Intersection

The command works very differently if you choose the *Intersection* button. In this case the options which are pertinent are those in the *Intersection* group. When calculating intersections, you can consider only the lines of the walls or all the lines which the drawn segment double line runs into. The *Intersection* button allows you to indicate a multiline which breaks lines at will. The command draws a horizontal, vertical or aligned dimension taking the projection on the dimension line of the intersections found as dimension points. Aside from the Intersection filter, you can exclude lines which you do not want to dimension by placing them on

frozen layers. The multiline path is free. It is possible to pass over the lines several times, to draw multiline sequences with no intersection and the multiline can be as long as you wish. The following example explains the DPLAN requests, once launched by pressing *Intersection*.

Command: DPLAN

Specify a point on the dimension line:

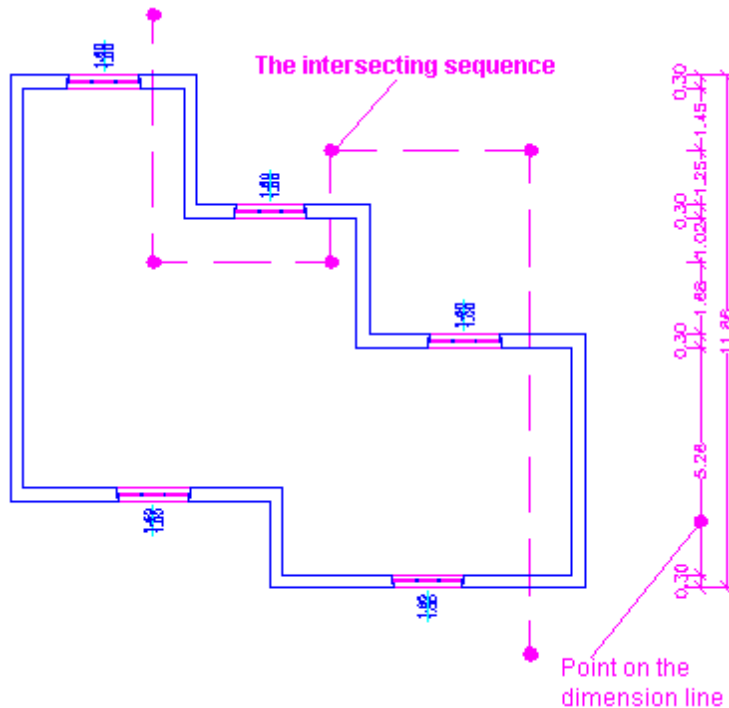
First point of the intersecting sequence:

[Undo]Point number 2:

.....

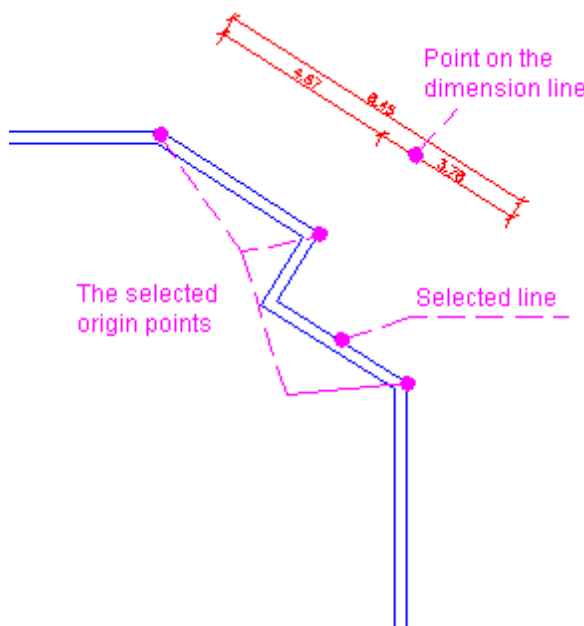
[Undo]Point number 7:

Do you want the overall dimension? (Y/N) <N>:



Dimensions for vertices

The DV command allows you to dimension a series of points in a set direction, such as the endpoints and midpoints of lines. The requests of the command are the following.



Command: DV

Dimension line angle, Enter if you want select a reference line or [layer and style Options]:

<ENTER>

Select a line parallel to the dimension line:

Specify a point on the dimension line:

Specify the first origin point:

■■■■■■■■

Do you want the overall dimension? (Y/N)

 $\langle N \rangle: Y$

To define the projection straight line of the points to be dimensioned and the direction of the dimension line, you can select a line or indicate an angle. You are then asked where the dimension line must pass. Last of all the points must be selected. You can select as many points as you wish.

After you have dimensioned the chain of

points, you will be asked whether you wish to have the overall dimension of the chain. The overall dimension will be moved with respect to the linked dimensions at a value set in *Overall dimension offset* taken from the [ADDPREF](#) dialog box.

Dimension layer and style

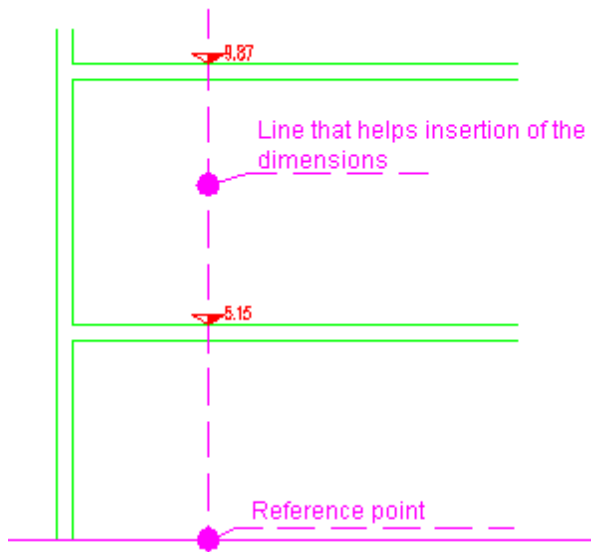
The layer used with this command is the same as the DPLAN command. The command has an option which allows you to choose both the layer to place the dimensions and the style.

The sizes of the dimensions

The style used is the one currently set. The size of the text and of the arrow symbols depends on the dimension style and the [plot scale factor settings](#).

Elevation views dimensions

The DELEV command automatically dimensions elevations. This command is especially useful to dimension sections and front views. The requests of the command are as follows.



Command: DELEV

Specify reference point or Enter to use the previous one:

Specify the dimension value of this point:
Specify the insertion point:

.....
The reference point and its dimension are the base for calculating the dimension values indicated with the following points. It is possible to confirm previously entered reference values. The command has the intersection snap to acquire points to be dimensioned. This is why it is convenient to draw a help line which also aligns dimensions vertically.

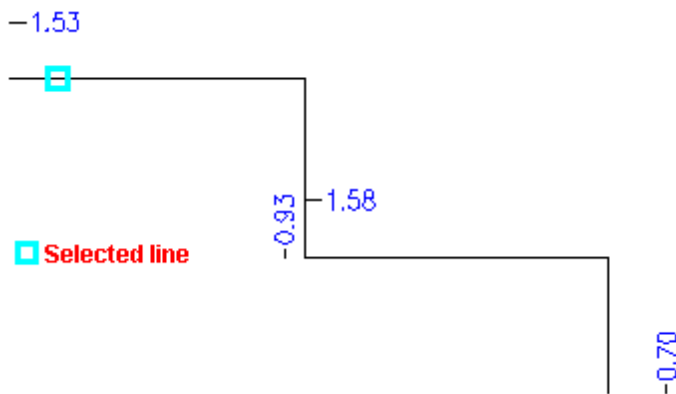
Layer

The layer used with this command is 2DELEV_DIM.

Stile and dimensions of elevations

These elevations are blocks inserted automatically in certain positions. The size of the text and symbols depends on the settings made in the [plot scale factor](#). You will see that when the scale factor changes, dimensions are not scaled automatically. However they can all be scaled with the AutoCAD property command since the point of insertion corresponds to the point to be dimensioned.

Multiline dimensions



This command is useful to show the dimensions of multilines. You can select a linked set of lines or a polyline. You must pay attention to the point where the line or polyline section is selected. By selecting close to one endpoint rather than the other, you can change the side where the dimension symbols are inserted. The settings used are those of the internal room dimensions dialog box. As for the internal room dimensions command, each time you can cancel

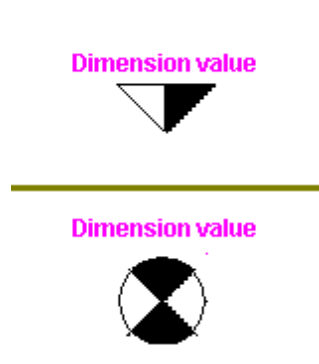
dimensions inserted automatically which you do not want to see.

Command: DBROKEN

Select a line of a multiline sequence to dimension:

Select dimension symbols to delete:

Dimension symbols



Two parametric objects can be recalled from the menus to enter dimension symbols in the 2D plan and in elevation. The only parameter is the numerical value of the dimension.

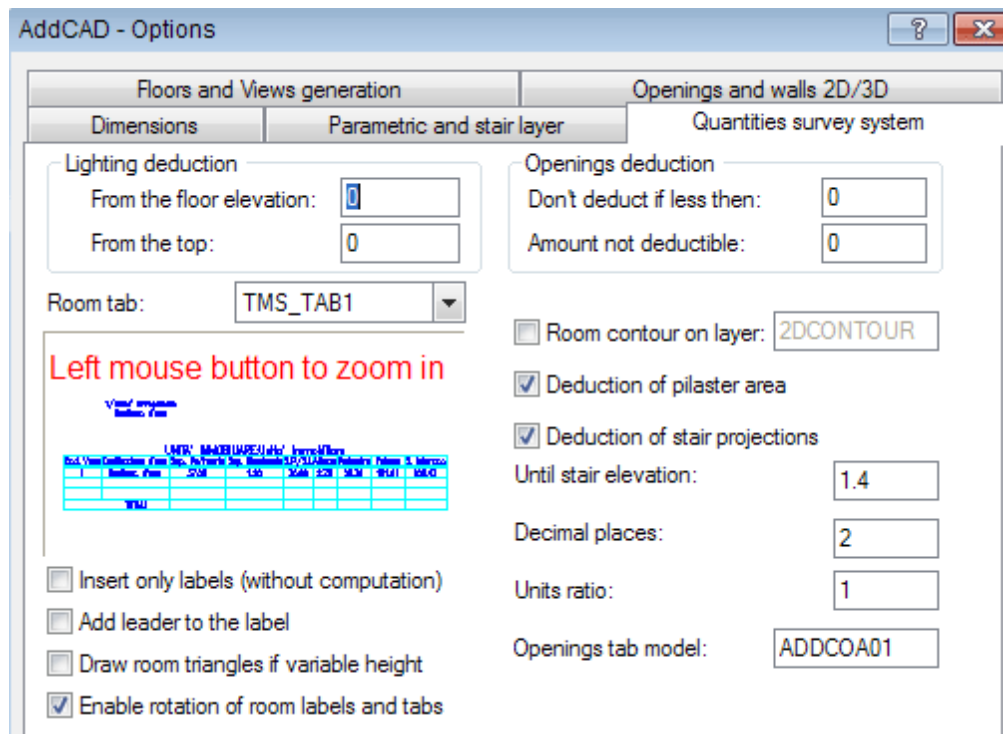
Layers used

The entities are placed in 2DDIMENSIONS__L and 2DDIMENSIONS__T, the symbol and text respectively. The block layer of the first symbol is ELEVATIONDIM while for the second symbol, the name is joined in accordance with the logic of belonging to a floor. The prefix will be ADDC and the suffix the floor elevation of the floor.

The sizes of the dimensions

The size of the text and symbols depends on the settings made in the plot scale factor. You will see that when the scale factor changes, dimensions are not scaled automatically. However they can be scaled with the AutoCAD property command since the point of insertion corresponds to the point to be dimensioned.

Survey system options



There is a specific tab of the AddCAD options dialog box containing quantities survey system options. From a first look we can see that it is possible to choose the room labels and tabs model. With the *Room tab* box, the list of all models can be opened and the model in the image below viewed

one by one.

Lighting deduction

These are standard light deduction values to be attributed to the windows. Values which will be subtracted from the height of the window to calculate the lighting area. The windows of the library take these parameters into account to [calculate the lighting area](#). The [LIGHT DEDUCTION command](#) allows you to edit the individual openings to assign specific deduction values.

Openings deduction

Don't deduct if less then indicates the minimum area an opening must have to be able to be deducted from plaster, painting, walls etc. surveys. *Amount not deductible* is the area of an opening which must not be deducted. The amount deducted from the wall area is therefore the opening area minus *Amount not deductible*. Of course as long as the opening area is larger than *Don't deduct if less then*.

Deduction of pillar area

The pillars inside the rooms can be deducted from the floor surface.

Deduction of stairs projections

The areas taken up by stairs up until a set height on the lower side of the staircase can be deducted from the floor surface.

Decimal places

This shows the accuracy used in writing the metric quantities survey system, openings tab and room tabs and labels.

Units ratio

If you wish to work graphically with a certain unit of measurement and print calculations with another, here you must write the ratio between the first and second.

Room contour on layer

When defining rooms, [contour polylines of the room](#) can be generated and placed on a certain layer.

Openings tab model

This is the [openings tab model](#) you wish to use.

Insert only labels

When room labels are inserted, room computations are done immediately. For plans with a small number of rooms (less than 15 per floor) the computation is instant. For more complex distributions, the time for computation increases exponentially. In these cases, you can first insert all the labels and then recalculate the entire floor.

Add leader to the label

If the room is very small, the label should be placed on the outside and a leader drawn indicating the interior of the room. Whether or not the definition path is generated depends on what was chosen during [room definition](#). Whether or not it is enabled in this dialog box only has the default value meaning.

Drawroom triangles if variable height

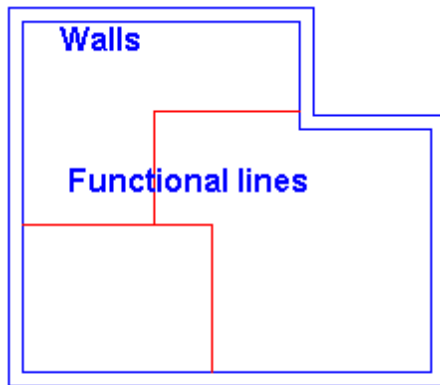
When calculating [variable height rooms](#), such as attics, the triangular breakdown can be drawn for each room calculated leading to the computation of the average weighted volume and height.

Enable rotation of room labels and tabs

When this option is enabled, together with the point of insertion, the rotation angle is also requested when inserting labels and tabs in the drawing. The rotation angle will be memorized to recalculate the rooms.

Functional areas

The FUNLINE command allows you to draw functional lines which divide large rooms into distinct functional areas. Functional lines are normal AutoCAD lines such as those of walls, with the particular feature of belonging to a layer with a special and reserved name. In fact the name is ADDC2DFUNL\$<FE> where <FE> stands for floor elevation. In conclusion, if we have drawn normal lines and wish AddCAD to interpret them as functional lines, we must bring them to that layer. From the [rooms computation](#) viewpoint, AddCAD considers the functional lines as room delimitations without the presence of a wall.



The FUNLINE command is similar to the WALL command, which however is much simpler to use. In fact, the requests are the following.

Command: FUNLINE

Specify start point or [Selection/Enter]:

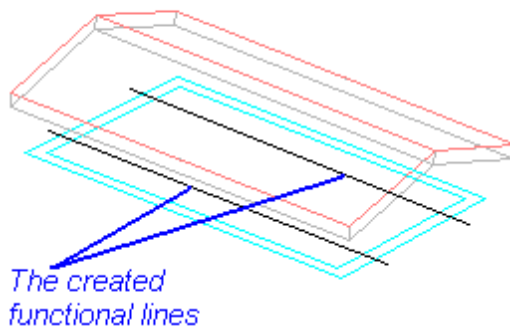
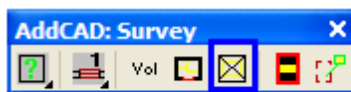
Specify end point or [Selection/Undo]

It is possible to select a wall or another functional line to be linked directly just like with the [WALL command](#).

The *Undo* option is always available to undo segments drawn incorrectly. The lines do not necessarily have to start from an endpoint of another line but can start from a midpoint. Obviously functional lines cannot cross wall segments. The figure has an

example of functional delimitations inside a large room.

Under-roof functional lines



Often areas underneath the roof with a minimum set height need to be isolated. This makes it possible to have the area corresponding to a height which exceeds a certain value. To achieve this result, just draw functional lines on the under-roof floor corresponding to that height.

The command then simply asks the height and draws the functional lines on the current floor elevation.

Command: ROOFFUNL

Elevation from floor where to project the functional lines<1.40>:

The functional lines are drawn wherever the roof slab soffit is at that height. In order to achieve a

delimitation of the actual areas (functional lines touching the walls) it could be necessary to cut the functional lines obtained.

Floor volume

The VFLOOR command allows you to calculate the volume and area of a floor. The command requests you to select a line or arc of the external perimeter. VFLOOR prints a message indicating the area and volume of the selected floor. The constant floor height is used to calculate the volume.

Command: VFLOOR

Select the external perimeter of the floor or of the room:

Response of command:

FLOOR ELEVATION:0

AREA=97.002

VOLUME=271.606 (AVERAGE FLOOR HEIGHT=2.80)

The conditions for a positive outcome of the command are the same as the calculation of room data. There must be a closed perimeter, made up of AddCAD parametric arcs, lines and openings.

Deduction of lighting areas

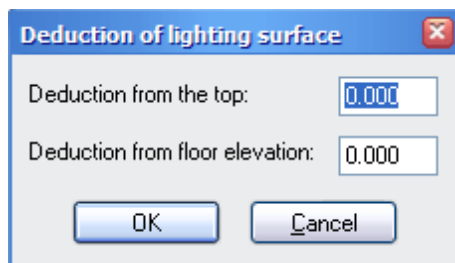
Windows are objects with lighting attributes and can take [deduction strips into account](#) due to the presence of balconies, parapets etc.

Let's suppose that we have a French door 2.60 m high and 1.20 m wide. The opening area therefore is $2.60 \times 1.20 = 3.12 \text{ m}^2$. It's not necessarily a lighting area as well.

Generally, the lighting area needs to be reduced by making deductions from the top and bottom. Let's suppose we assign the previous opening with 50 centimeters on the bottom strip and 30 centimeters on the top strip which are not lighting.

The lighting area will be $(2.60 - 0.50 - 0.30) \times 1.20 = 2.16 \text{ m}^2$.

For windows with windowsills, the total deduction due to the bottom strip is calculated as the net height of the windowsill. Standard deduction strips for all windows are set in [Quantities survey system Tab](#) of AddCAD options. It is possible to individually change the amount of the deduction strips for the windows. If you wish to change the value of the deduction strips individually, use the LIGHTDEDUCTION command. LIGHTDEDUCTION asks you to select the openings where the light deduction must be changed and shows the following dialog box.



Command: LIGHTDEDUCTION

Select windows to which you want change the lighting deduction amount:

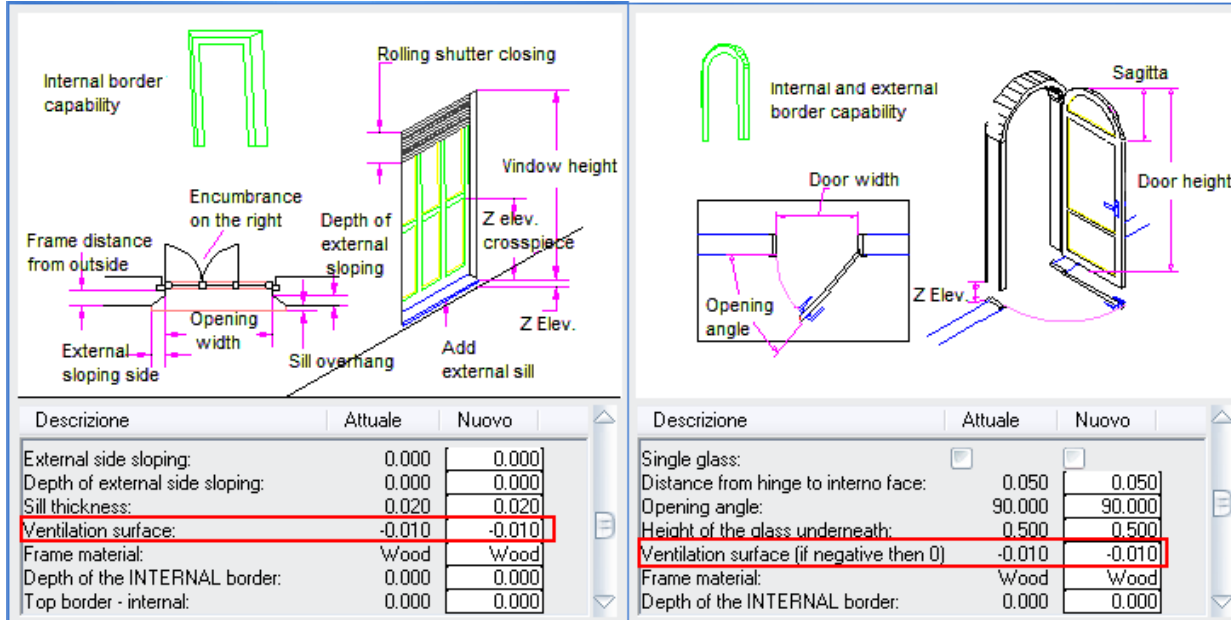
If openings with different light deduction values are selected, the edit field displays the message 'VARIES'. You can modify the values and exit by pressing OK to update the deduction data.

Ventilation surface parameter

Most building regulations require this data to be calculated for those rooms where a minimum value is required. There are room tabs and frame tabs which manage this type of data. The presence of a specific parameter is necessary because the presence of glass in a frame does not always mean that its surface is ventilating as well. It depends whether the window can be opened or if it is fixed. This is not important for the lighting surface.

Therefore to correctly manage the ventilation surface, openings have a parameter which has nothing to do with their graphical appearance. This parameter must allow us to

manage all situations with just one numerical value. This speeds up management of this information.



The image shows two diagrams illustrating window and door parameters, and two corresponding data tables.

Left Diagram (Window): Shows a window with various parameters labeled: Internal border capability, Rolling shutter closing, Window height, Encumbrance on the right, Depth of external sloping, Z elev. crossplace, Z Elev., Frame distance from outside, Opening width, Sill overhang, Add external sill, External sloping side, and External side sloping.

Right Diagram (Door): Shows a door with various parameters labeled: Internal and external border capability, Sagitta, Door height, Door width, Opening angle, and Z Elev.

Left Table (Window Data):

Descrizione	Attuale	Nuovo
External side sloping:	0.000	0.000
Depth of external side sloping:	0.000	0.000
Sill thickness:	0.020	0.020
Ventilation surface:	-0.010	-0.010
Frame material:	Wood	Wood
Depth of the INTERNAL border:	0.000	0.000
Top border - internal:	0.000	0.000

Right Table (Door Data):

Descrizione	Attuale	Nuovo
Single glass:	<input type="checkbox"/>	<input type="checkbox"/>
Distance from hinge to interno face:	0.050	0.050
Opening angle:	90.000	90.000
Height of the glass underneath:	0.500	0.500
Ventilation surface (if negative then 0):	-0.010	-0.010
Frame material:	Wood	Wood
Depth of the INTERNAL border:	0.000	0.000

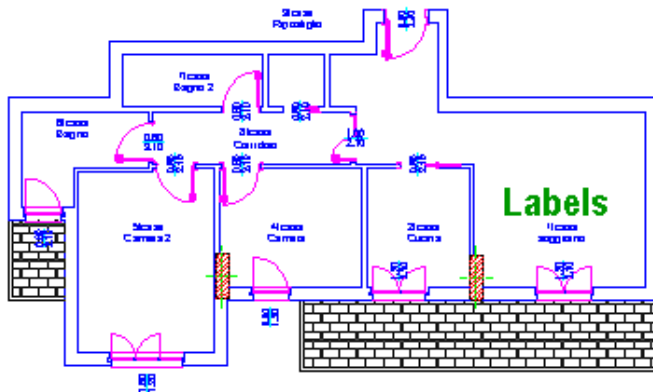
Practically it is possible to write a positive value to specify a surface according to the user's requirements or zero (0) so that the opening being inserted is not considered a ventilation surface. If a negative value is left or written, whatever it may be, the ventilation surface will be the one calculated by the program which is usually equal to the opening in the wall. Sometimes, as for inside doors or glass windows which do not seem to open, the value indicated is negative, in this case, the value calculated by the program as 0.

Room data calculation

Introduction

While you are drawing building plans, you can have the project data under control. For example, the [overall volume](#) of a floor can be kept under control or you can [view the data of an individual room](#). Some sizes characteristic of rooms can be calculated and organized in specific tables. Data is calculated for the rooms and in some cases the tables are set up to contain the overall values of the floor or dwelling unit. Instead of complete tables, room labels can be inserted which describe some features of the room. Label models can be created with or without tables. Many models are default and are included in AddCAD.

Once the project finished, tables can be inserted for openings. Formats are available for door, window and generic opening tables. The room data and the openings tab data can be taken from other programs in readable files.

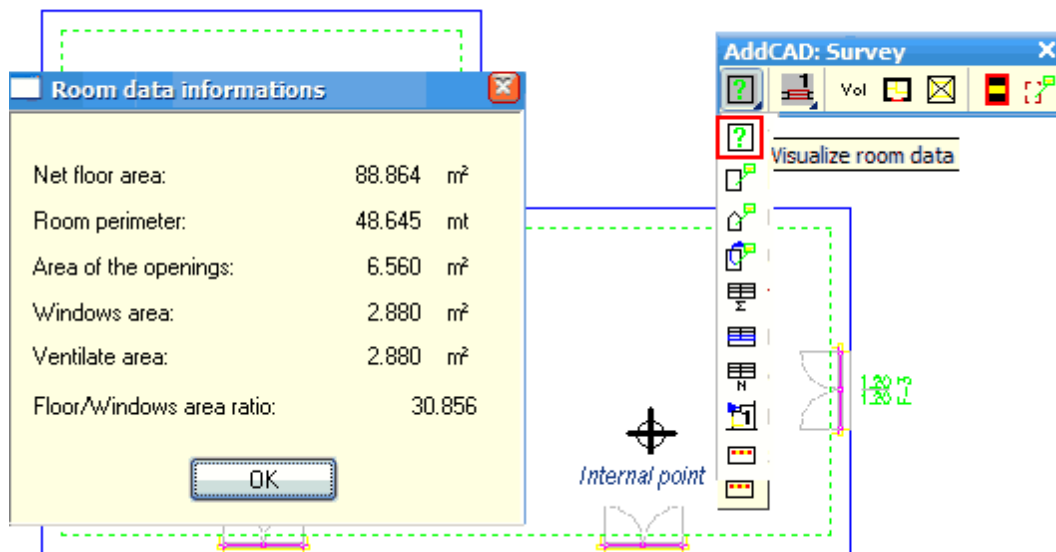


Table

Col	Yano	Descrizione data	Sup. Pavedimen	Sup. Illuminazione	S.P.E.I.	Altezza/Pavimento	Volume	S. Insieme
1		Bagno 2	22.37	2.32	0.00	2.70	60.40	81.73
2		Cucina	8.25	2.32	2.32	2.70	10.10	17.19
3		Pigiama	1.58	0.00	AAA	2.70	0.00	4.21
4		Corridoio	3.82	0.00	0.00	2.70	11.90	20.20
5		Bagno 2	12.20	1.92	0.00	2.70	14.80	24.70
6		Bagno	4.25	1.88	2.20	2.70	10.20	19.10
7		Bagno 2	9.24	0.00	AAA	2.70	3.20	10.97
8		Corridoio	5.40	0.00	AAA	2.70	11.40	14.50
9		TOTALE	88.86	9.80			98.40	179.11

Room information

The INFOUNIT command allows you to visualize the data of a room by selecting an internal point. No room tab needs to be created to use the command. This function is useful during the designing stage to check some fundamental room data. Like for the insertion of rooms tags, you are requested to indicate the internal point of a closed room. The following figure shows the contents of the report of the selected room and the button on the toolbar to launch the command.



Room data calculation

The SPACEUNIT command allows you to define rooms with the possibility of inserting relative labels. The command also updates the room tab if the label model being used. The calculation procedures are automated, you just need to select an internal point of the room to be calculated. In many aspects the behavior of the command depends on the

[label model](#) and the settings in *Quantities survey system tab* of AddCAD options. The label model can be chosen in the *Quantities survey system tab*. The program supplies a wide range of [label](#) and [tab](#) models.

First of all the command displays the dialog box to acquire data which cannot be calculated and must be indicated by the user. One example of data which cannot be calculated is the room name. The dialog box depends on the label model in use and on the command used to define the room.

Once the data has been entered in the dialog box, the program looks to see whether a tab related to the calculation being made is already present. If the tab does not yet exist, it requests the parameters and its point of insertion. Usually you need to indicate the number of rows in the tab.

Command: SPACEUNIT

Pick internal point of the room of which calculate data or [Polyline]:

Rotation angle <0.0>:

Using polylines as rooms

In order to define rooms, you may select a point inside a room or else some closed polylines. As a matter of fact while the SPACEUNIT command is selecting the internal point, the *Polyline* option can be enabled allowing you to select a polyline. After having selected the polyline, everything goes ahead as if you had selected the internal point of a room. This option is used to calculate and insert room data in tabs without actual masonry delimiting the areas. Something similar is possible with functional lines. This has the advantage of drawing free room shapes which are not based on lines. The polyline will be attached to the room label created during the room definition stage. Therefore if the polyline is changed, for example stretched, and the room data is recalculated, the data of the polyline is recalculated as well and consequentially the label and tab updated.

First internal point in room or polyline

In order to position the label, the program asks you to indicate two points. The first point must be inside the room being calculated or the selected polyline.

Second point and label leader generation

The second point is useful if you wish to insert the label outside of the room. In this case, you can generate a linking leader between the first and second point. If you press *Enter* when asked for the second point, the label will be inserted in the first point. The leader will only be drawn if the relative option has been enabled in the [room definition window](#).

Conditions for a successful outcome of the room data calculation

The program calculates the data, inserts the label and updates the tab if this is foreseen. The following conditions must be kept in mind for a successful outcome of the calculation. Only parametric objects of the library supplied with AddCAD or which meet the requirements of the openings definition must be used for [openings](#) and [pillars interacting with walls](#). Whether or not the lines of the openings are broken does not matter for the command. The objects must not have been exploded. The current room elevation must be that of the room. The room perimeter is calculated by searching for a path which links the endpoints of the lines which can only be broken by openings and pillars. As far as pillars are concerned, the perimeter calculation takes into account the shape of the pillar by considering its plan encumbrance polygon. Fundamentally the program places itself at the

line closest to the label point of insertion and tries to go back there through a continuous path made up of lines, arcs, openings and pillars. If it does not find this path, the program is not capable of calculating the desired quantities correctly. In this case, the maximum path calculated will be highlighted in red. There was definitely a problem of continuity or overlapping at the endpoints of this path which can be analyzed and corrected with the [ADDAUDIT command](#). If the path was found, the closed path is highlighted in green so that the result can stand out better. The highlighted sections can be restored to their normal status with AutoCAD's REDRAW command.

Opening surfaces and lighting surfaces are obtained from the frame definition data. For example windows are all considered lighting whereas doors are not.

Surface values imposition

During the room definition stage, the floor, ventilation and lighting surface values can be preassigned by hand. If one of these surfaces has been assigned by hand, the program accepts these values as true without proceeding with calculation.

Floor, plaster and lighting area deductions

The program can perform a series of deductions of the calculated areas based on the options set in the *Quantities survey system tab*. It is possible to deduct floor areas taken up by pillars and stairs. It is possible to set the usual total volume plaster deduction logic and it is possible to deduct lighting area strips from window areas.

Height and style of texts

The text height of the labels and tabs is controlled by the [SCALEF command](#). To generate tabs, AddCAD uses two text styles, one for the descriptive row and one for the rows of data and for the labels. The names of these styles must be written in the *Parametric and stairs layer tab* of AddCAD options. The definition of these styles can be changed at will.

Scale factor of label block

A height or width scale factor can be assigned to individual labels so that they can be adapted to the size of the rooms if there is little space. Any regenerations and recalculation take this scale factor into account.

Layers generated by inserting labels and tabs

The layers created by the insertion of labels can depend on the contents of the definition files. The models supplied with the program create the following layers.

2DROOMDATA Texts of labels and tabs

2DROOMDATA_L Lines of labels and tabs

ADDC2DROOMDATA\$<FE> Block layer of labels where <FE> stands for Floor Elevation.

2DROOMDATA Block layer of tabs

As you can see, the labels are objects linked to the [floor logic](#), while the tabs are not. They represent dwelling units and consequentially are not bound by floor logic.

Data for variable height room

The SUNITV command, like the SPACEUNIT command, calculates room data and inserts a label inside. Therefore see the page dealing with the [SPACEUNIT command](#) to know how this command. The difference between the two commands is that SUNITV considers the presence of roofs above the room. Some computation data for variable height rooms depends on the roof system present in the drawing. In particular it properly calculates the

room volume, the plaster surface of variable height walls and the average height provided by the ratio between floor volume and surface. When using this command, the dialog box for acquiring data does not request the room height.

Conditions for working properly

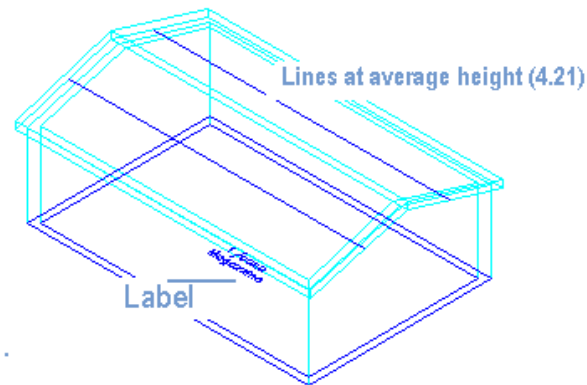
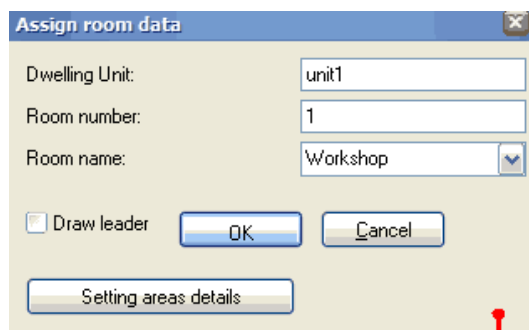
For the command to properly process data, the room must be covered by AddCAD roof elements (roof slabs and floor slabs) and the roof slabs must not overlap. You can always go back to this situation. As most the roof vertices must be stretched temporarily while calculations are being made.

Different kinds of labels

The labels inserted with this command are the same as those inserted with SPACEUNIT. The only difference is that they have been inserted to calculate variable height room data. This information is memorized together with the label. Thus it is possible with [SUNITR to recalculate all the data](#) of the labels, for each type of calculation for which it was inserted. If a room was at a constant height and it becomes at variable height, or vice versa, the label must be removed and inserted with the appropriate command.

Checking Average Height value

A simple way of checking where the average height room points are is using the [ROOFL command](#) which traces lines at the requested z elevation. The lines which represent the place of the average height points are placed on the HELPLINES layer, and therefore they can be deleted using the delete layer command. This check provides us with a quality visual feedback of the correctness of the calculation.

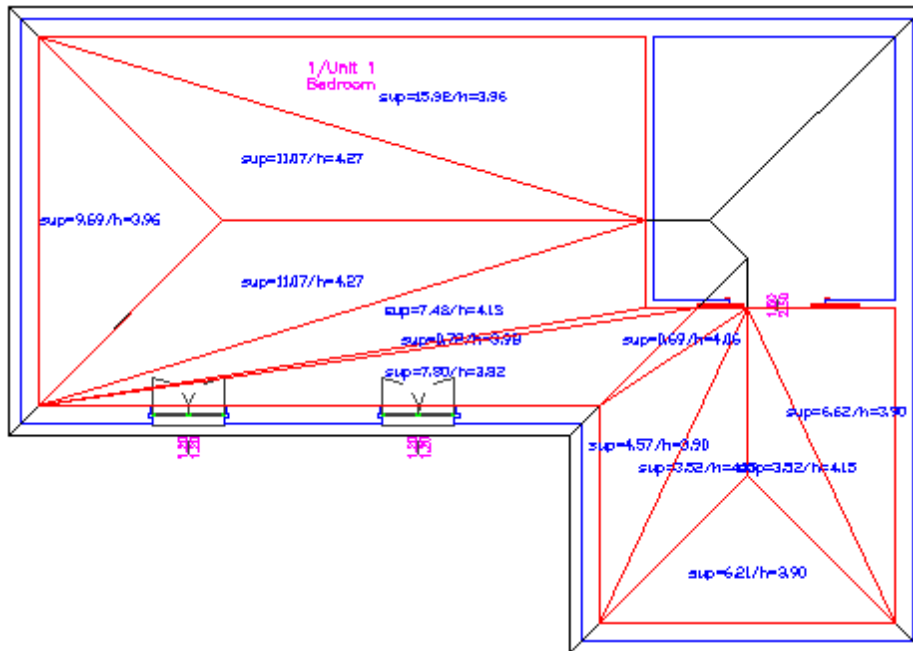


Dwelling Unit: unit1								
Room Nr.	Room name	Net floor area	Windows Area	W.A./N.F.A	Height	Perimeter	Volume	Plaster area
1	Workshop	22.05	2.89	0.13	4.21	19.00	95.23	81.33
Total								

Drawing triangles which individually make up the volume and average height calculation

During calculation of variable height room data with this command, it is possible to draw the triangulation used for the calculation on another layer. Each triangle is related to a room referring to an overlaying roof slab. The surface of the triangle itself and the average height are visualized inside each triangle.

The result is shown in the figure.



The layer used for the triangles and the measurements inside is called `ADDC2DROOMTRIANGLE$<FE>`.

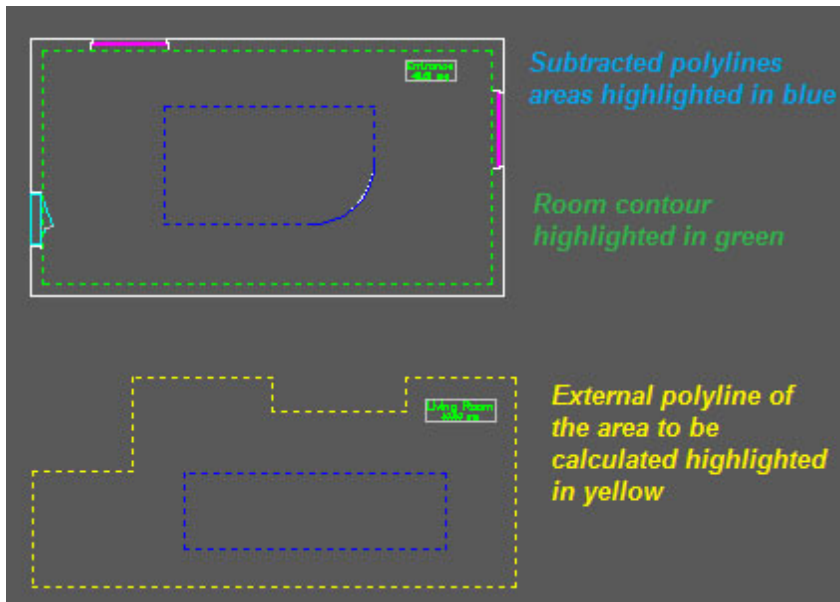
As can be seen the name follows that of the floor elevation logic. Practically the triangle layout can be plotted together with the plans demonstrating the result obtained.

To draw the triangles, select the relative option in *Quantities survey system tab*.

Polylines detracton

You can defined, by means of a closed polyline, any area to be deducted from the quantity calculation inside a room or inside a polyline of an area to calculate. The polylines to be deducted are automatically recognized and their contour is highlighted with blue color just like the program does for [stairs and pillars](#).

The `POLYDEDUCT` command allows you to assign and remove the feature of deductibility of polylines. Obviously the condition for the correct deduction is that the areas bounded by the polylines are internal to the room area or to the polyline of an external contour.



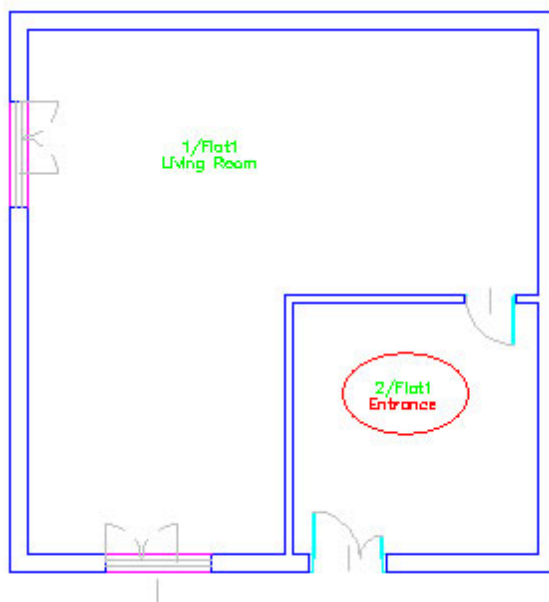
Command:
POLYDEDUCT

Select a closed polyline to which you want add deductibility[Remove deductibility]/Enter to finish:
<Select>

Select a closed polyline to which you want add deductibility[Remove deductibility]/Enter to finish:
<Enter>

As you can see, you can also remove the deductibility feature of polylines. By choosing the *Remove* option, the command prompts you to select the polylines to which you want to remove the deductibility.

Highlighting room with fixed areas



It is possible to associate values for floor, lighting and ventilation surfaces to individual rooms.

The SHOWTAGFIX command visualizes the rooms which have had areas fixed manually. In the figure, the *Entrance* has one of the surfaces assigned manually.

The command also restores normal visualization.

Command: **SHOWTAGFIX**
Enter to highlight room labels with fixed areas or [Highlight/Normal]:

The Normal option restores the room name with the usual color.

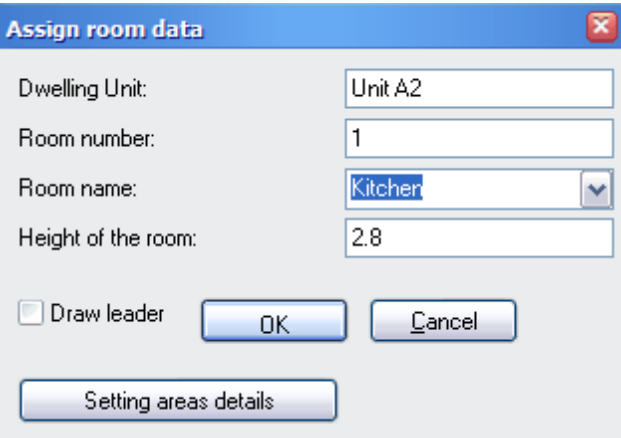
In order to highlight the room name with a different color, the command uses a specific layer called 2DROOMDATA_LABEL.

and exit by pressing OK.

Changing room tab name

The CNAMETAB command allows you to change the name of a room tab. This will also change the name of the dwelling unit since the name change spreads to the room labels which refer to the tab. You are requested to select the tab to be renamed and the new name.

Modifying descriptive room data

A screenshot of a software dialog box titled "Assign room data". It contains four input fields: "Dwelling Unit:" with the text "Unit A2", "Room number:" with the text "1", "Room name:" with a dropdown menu showing "Kitchen", and "Height of the room:" with the text "2.8". Below these fields is a checkbox labeled "Draw leader" which is unchecked. At the bottom are three buttons: "OK", "Cancel", and "Setting areas details".

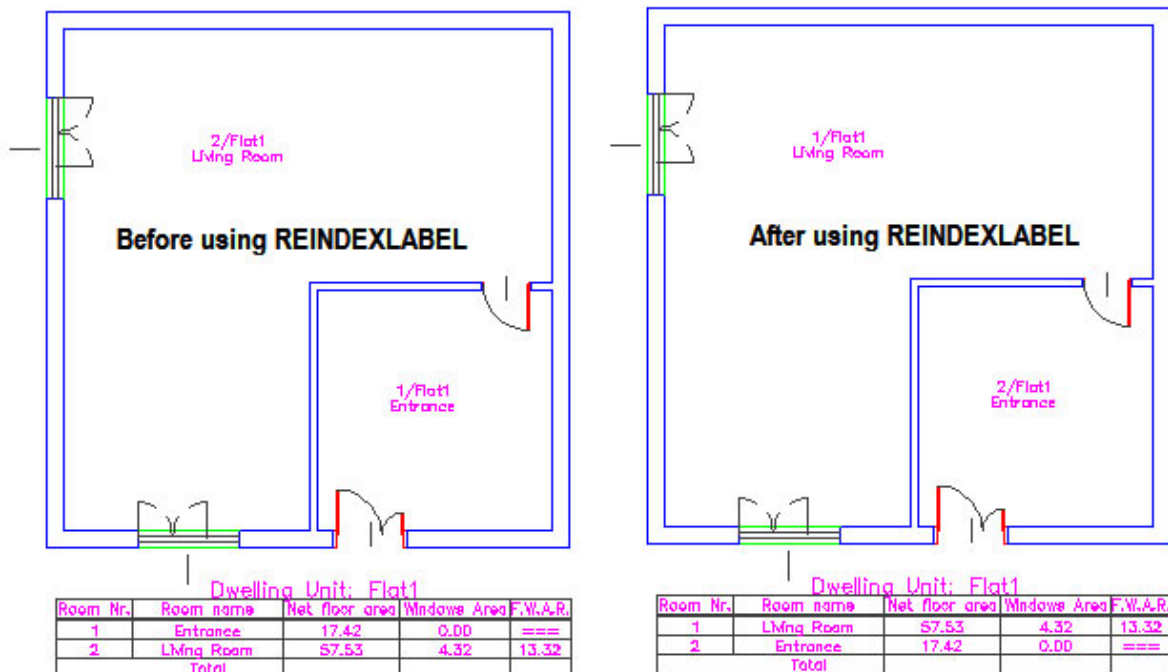
It is possible to modify the descriptive data of rooms without performing any calculations. This command is very useful in large projects with many rooms. With just one command it is possible to modify the room data assigned by the user and then, if necessary, to [recalculate all the rooms](#) of the floor and to update labels and tabs.

Command: MODTAG

Select a room label or Enter to exit:

Reassigning room numbers

Once the [rooms have been defined](#) by inserting the tabs in the relative rooms of the dwelling unit, a simple command can be used to reassign room numbers. This operation is especially useful when you wish to change the order of the rooms inserted in the tab. The example in the following figure shows a simple case of two rooms and the relative tab.



You can see in the figure how after the command has been used, besides having associated the rooms with a different number, their position is also updated in the room summary tab.

The requests made by the command regard the initial room number which will then be applied to the first label selected. Afterwards, you are asked sequentially to select the other labels in ascending order.

Command: REINDEXLABEL

Specify the initial room number for progressive renumbering:1

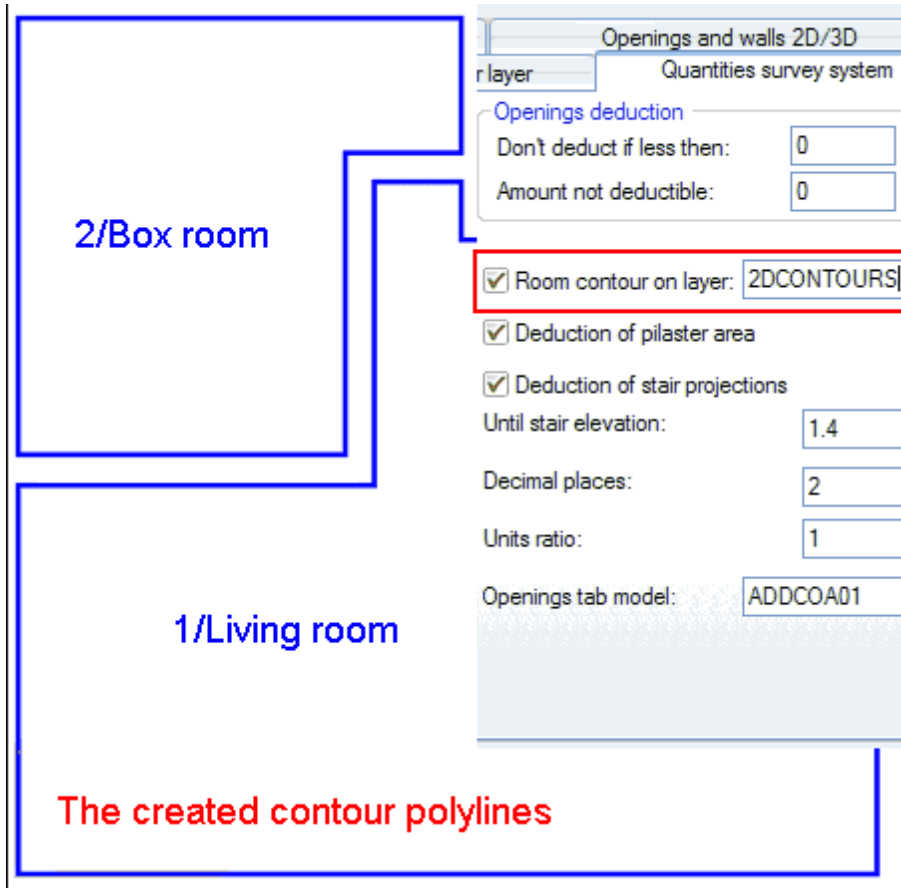
Select the room label to which assign the room number 1. Enter to exit:<select label nr. 2>

Select the room label to which assign the room number 2. Enter to exit:<select label nr. 1>

Select the room label to which assign the room number 3. Enter to exit:<ENTER>

Generating polylines of rooms

It is possible to generate the polylines of the perimeter contours of the rooms when the rooms are calculated. These polylines are updated automatically when room data is recalculated ensuing modifications to the plan. This feature is necessary in some building management applications. The figure shows where the contour polyline generation can be enabled. You must open the *Quantities survey system* tab of *AddCAD options*. The polylines are normally placed on the 2DCONTOURS layer, though it is possible to change layer by writing a different one.



Room data extraction

It is possible to extract room data to allow it to be processed by other programs by using the **EXTRACTR** command. The structure of the extraction file can be defined inside the *AddCAD.dex* file.

Command: **EXTRACTR**

The command requests the filename where the room data will be saved. The data of all the room tabs is extracted and therefore of all the dwelling units present in the drawing. Each row follows the record path specified in the *AddCAD.dex* format definition file.

Tabs and labels

Tabs and labels

Assign room data

Dwelling Unit: Flat1

Room number: 2

Room name: Entrance

Height of the room: 2.800

☐ Draw leader

OK Cancel

Fixed surfaces details

Assign room data

Dwelling Unit: Flat1

Room number: 2

Room name: Entrance

Height of the room: 2.800

☐ Draw leader

OK Cancel

Hide details

☐ Fixed floor area: 0

☒ Fixed windows area: 3

☐ Fixed ventilation area: 0

In order to calculate the room data, it is [possible to choose](#) from various label models. A label model can be associated to a tab and a tab can be associated to a total record. Therefore there are [label models without tab data](#) and [with tab data](#).

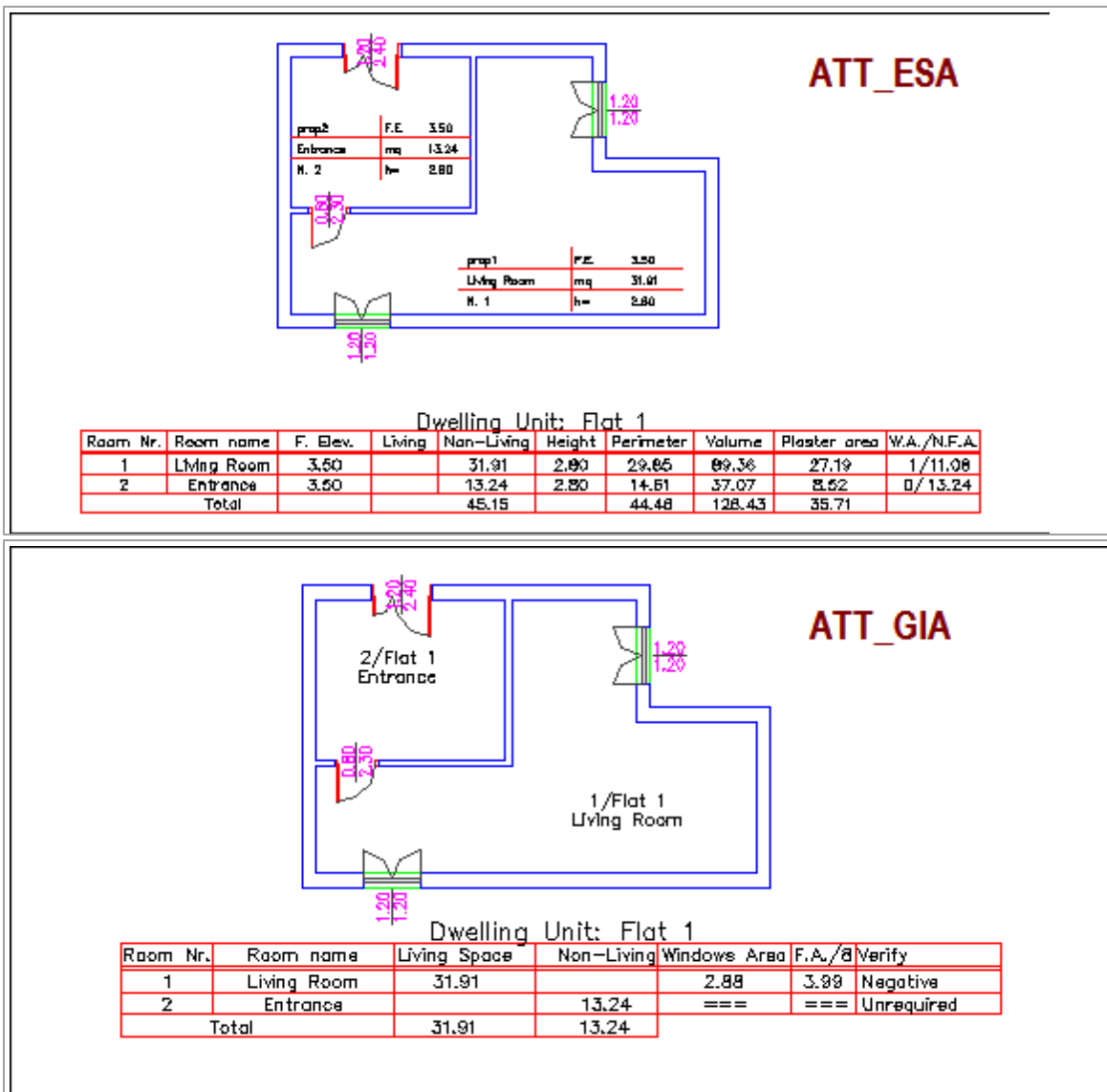
The dialog box for inserting room information can change by choosing different label models. If the label model also has a tab, then the dialog box will have references both to the tab and to the row of the tab.

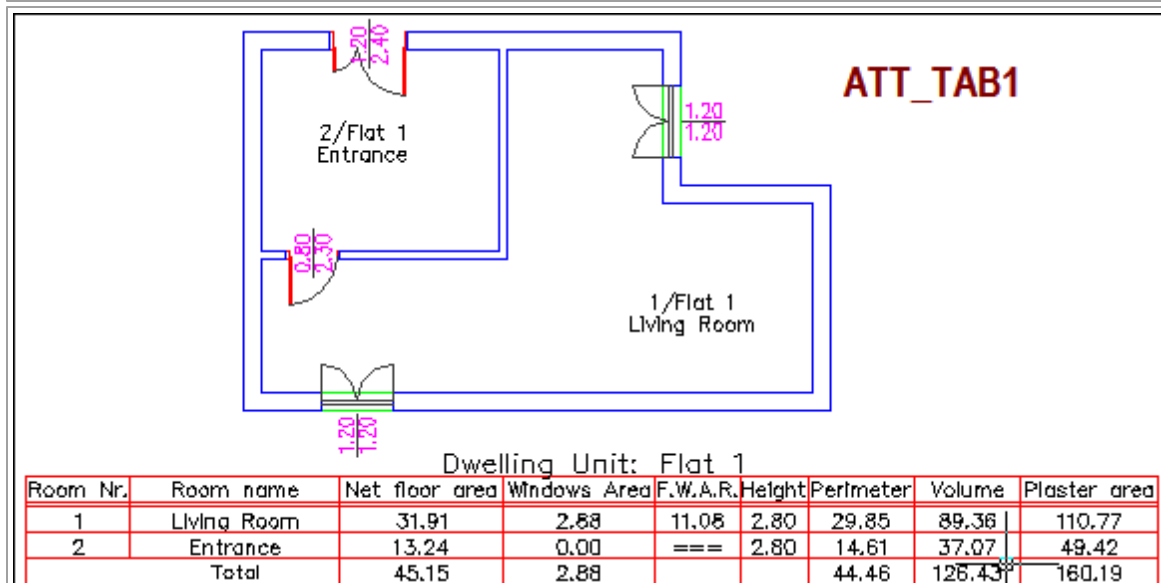
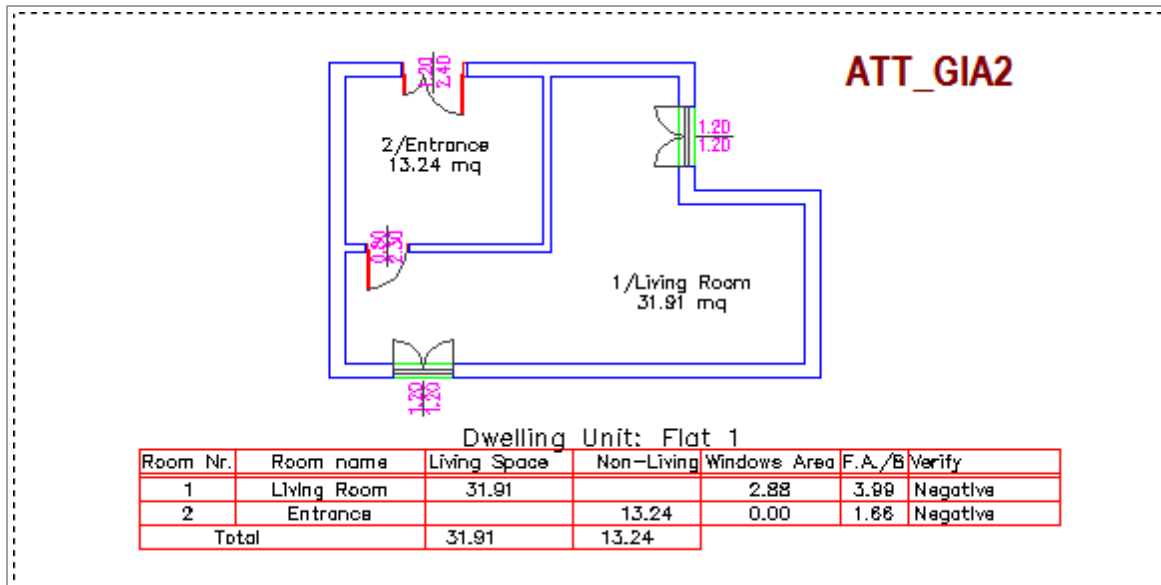
All the room definition dialog boxes have the *Draw leader* option. If enabled, it draws a leader line between the internal point of the room and the label insertion point. If a variable height room is defined, the *Height of the room* field is not proposed even though the label model provides it as information.

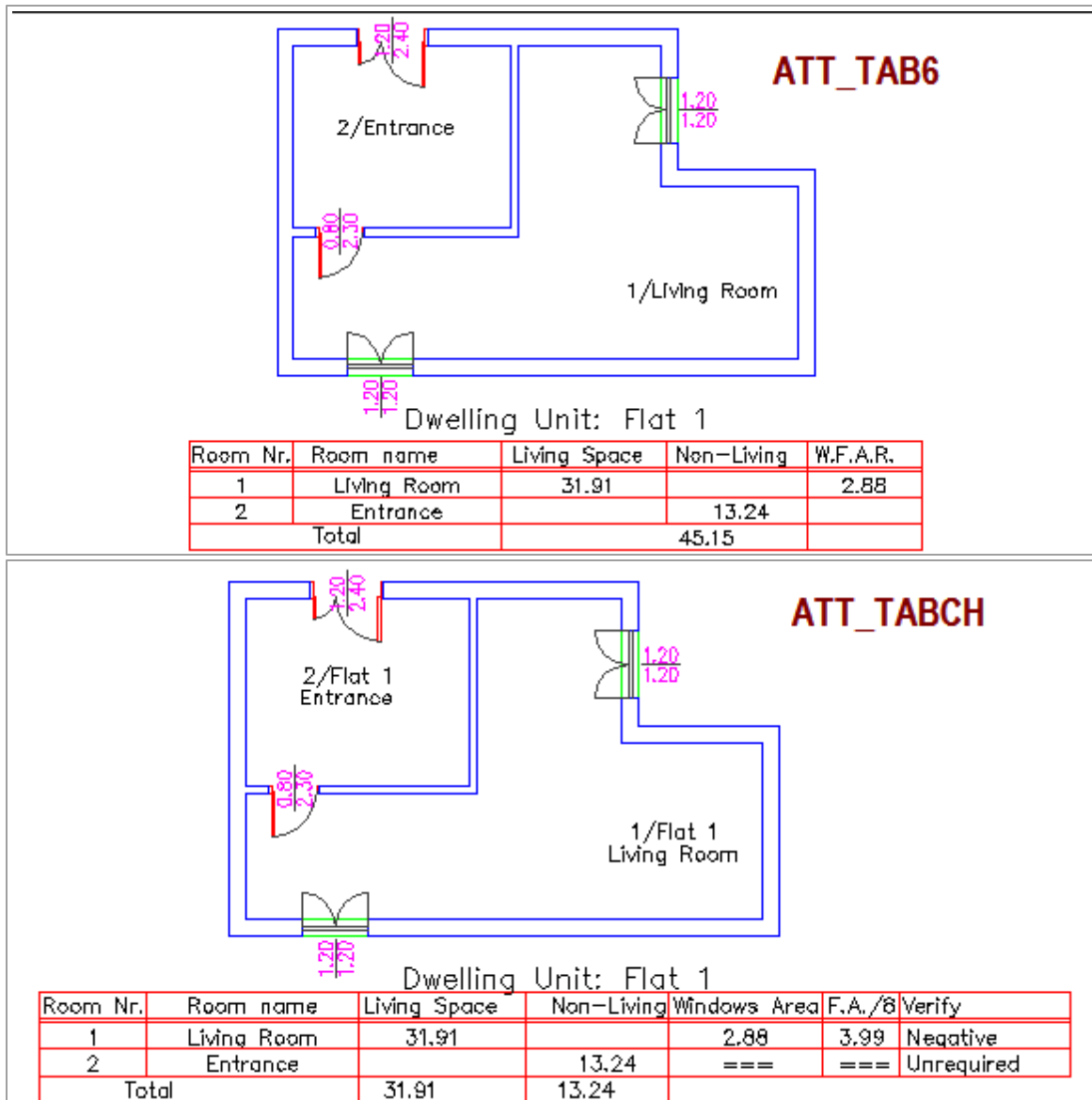
All room definition dialog boxes have a *Fixed surfaces details* button, which if selected makes it possible to decide which surfaces to impose a value by hand. In this case, the program will not perform automatic calculation for the selected surfaces. The data set is treated as if it were calculated by the program. At any time the room labels for which a surface was set can [be highlighted](#).

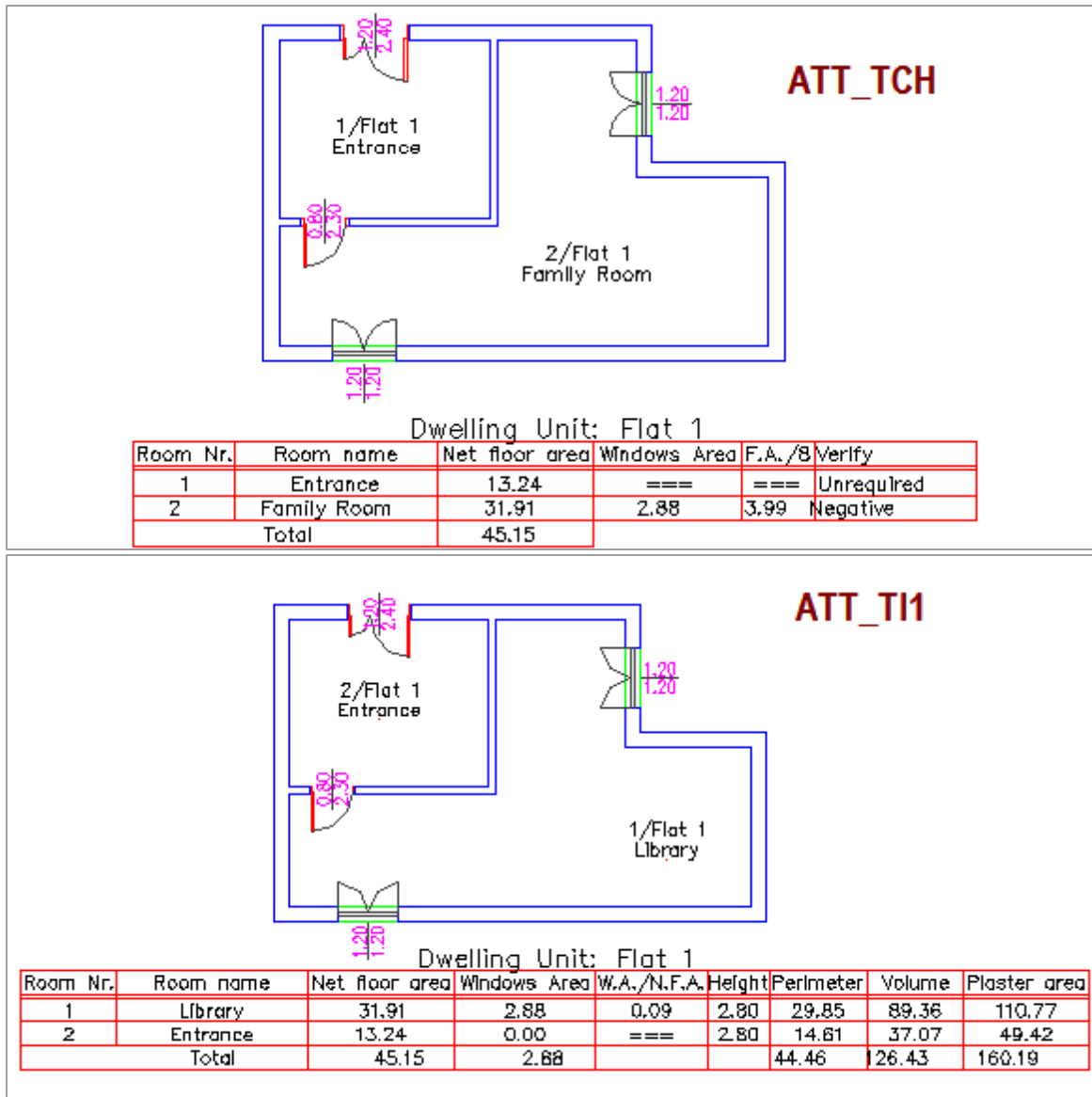
Tabs and labels are particular parametric

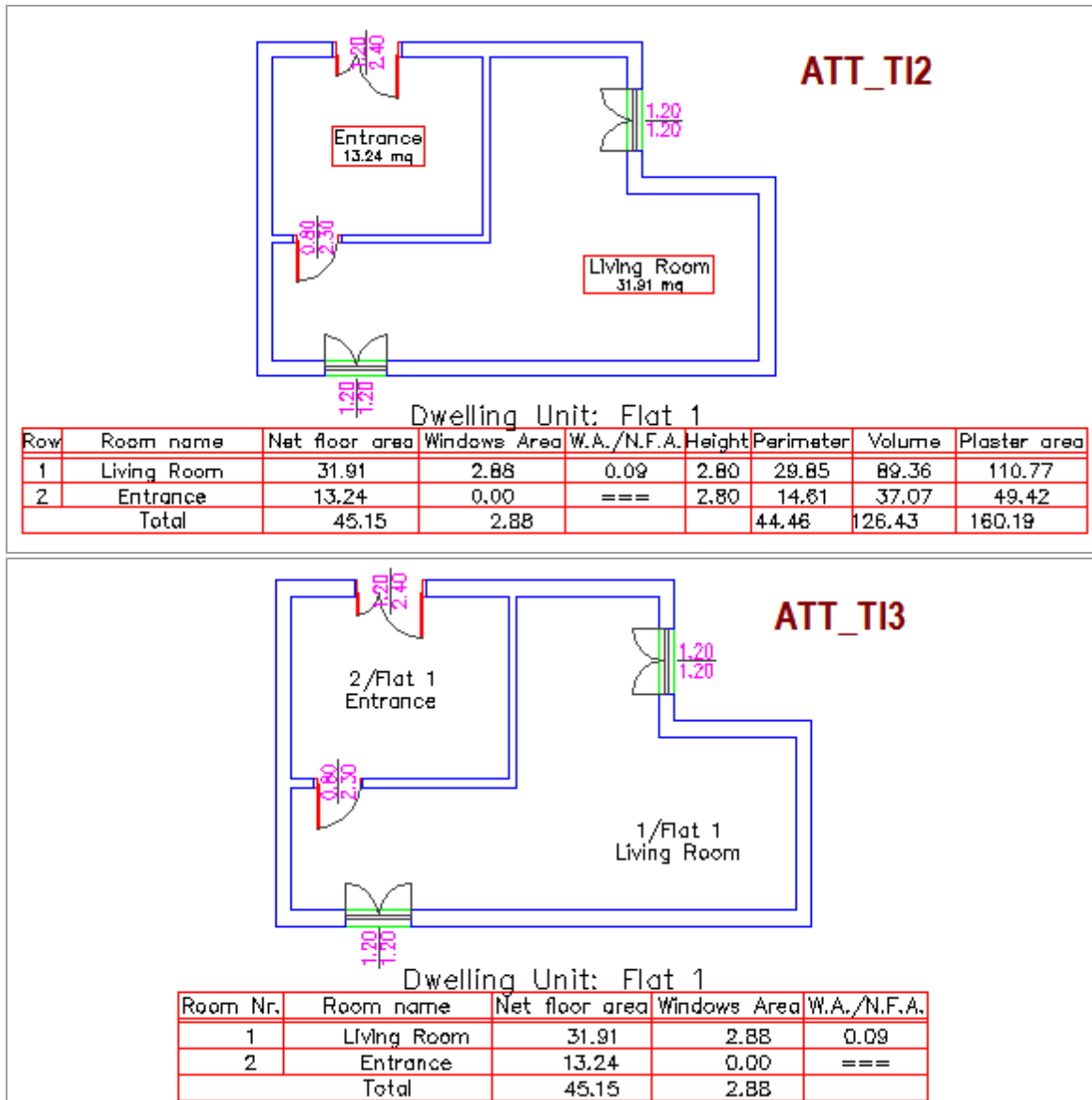
objects. The texts of the tab and label rows are attribute blocks. They can therefore be modified with AutoCAD's DDATE command. If the rooms are recalculated automatically, the values of the attributes are updated automatically as well.

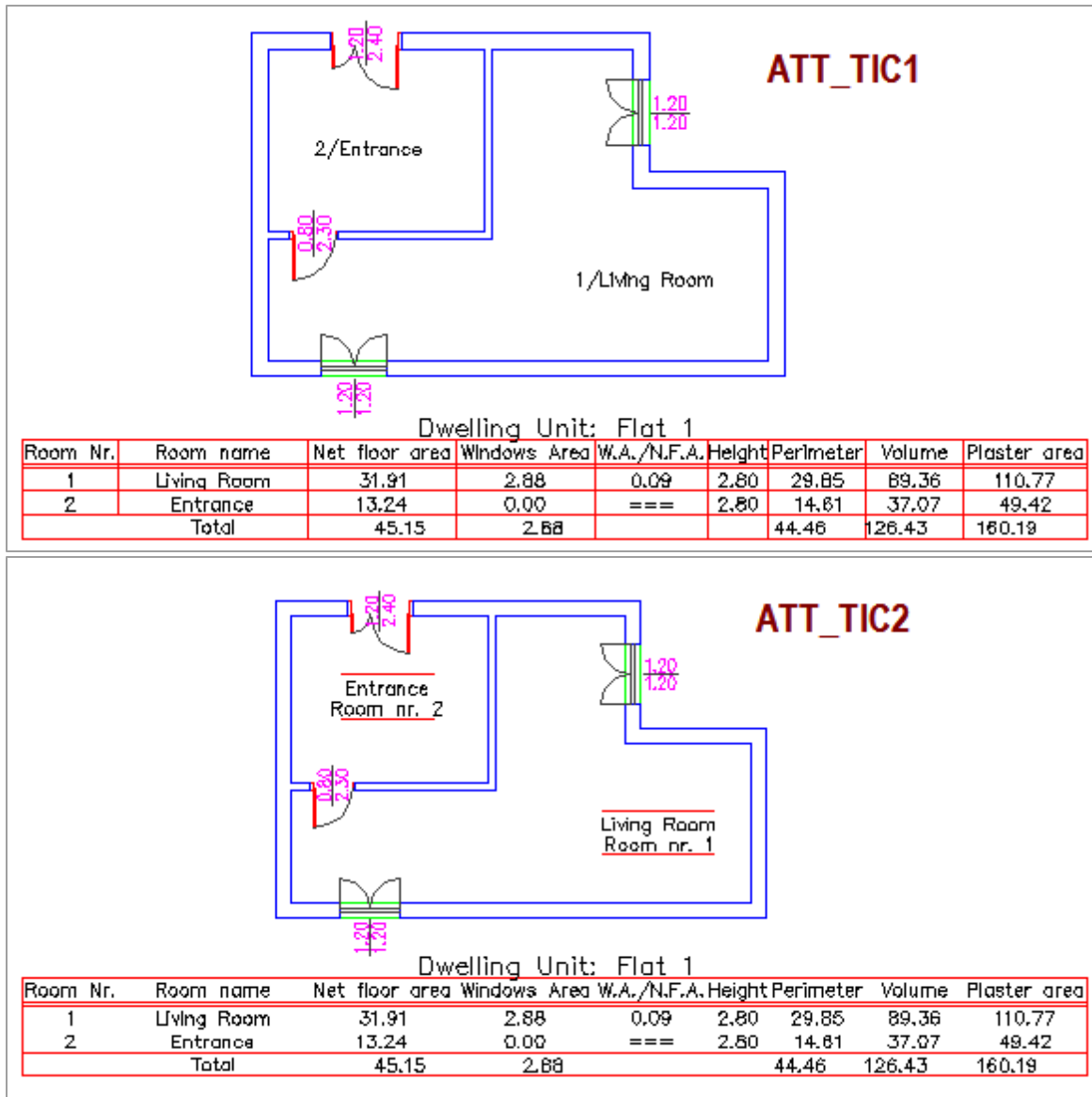
Room label with tab

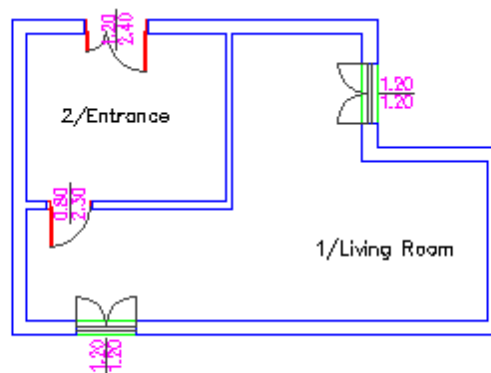








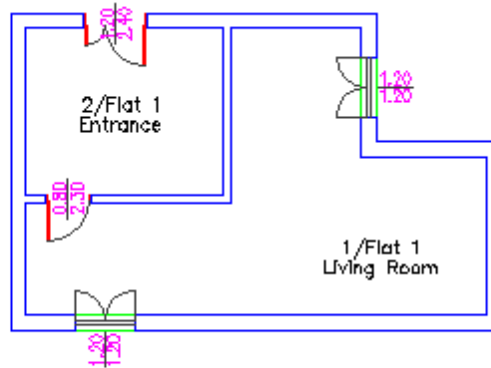




ATT_TIC3

Dwelling Unit: Flat 1

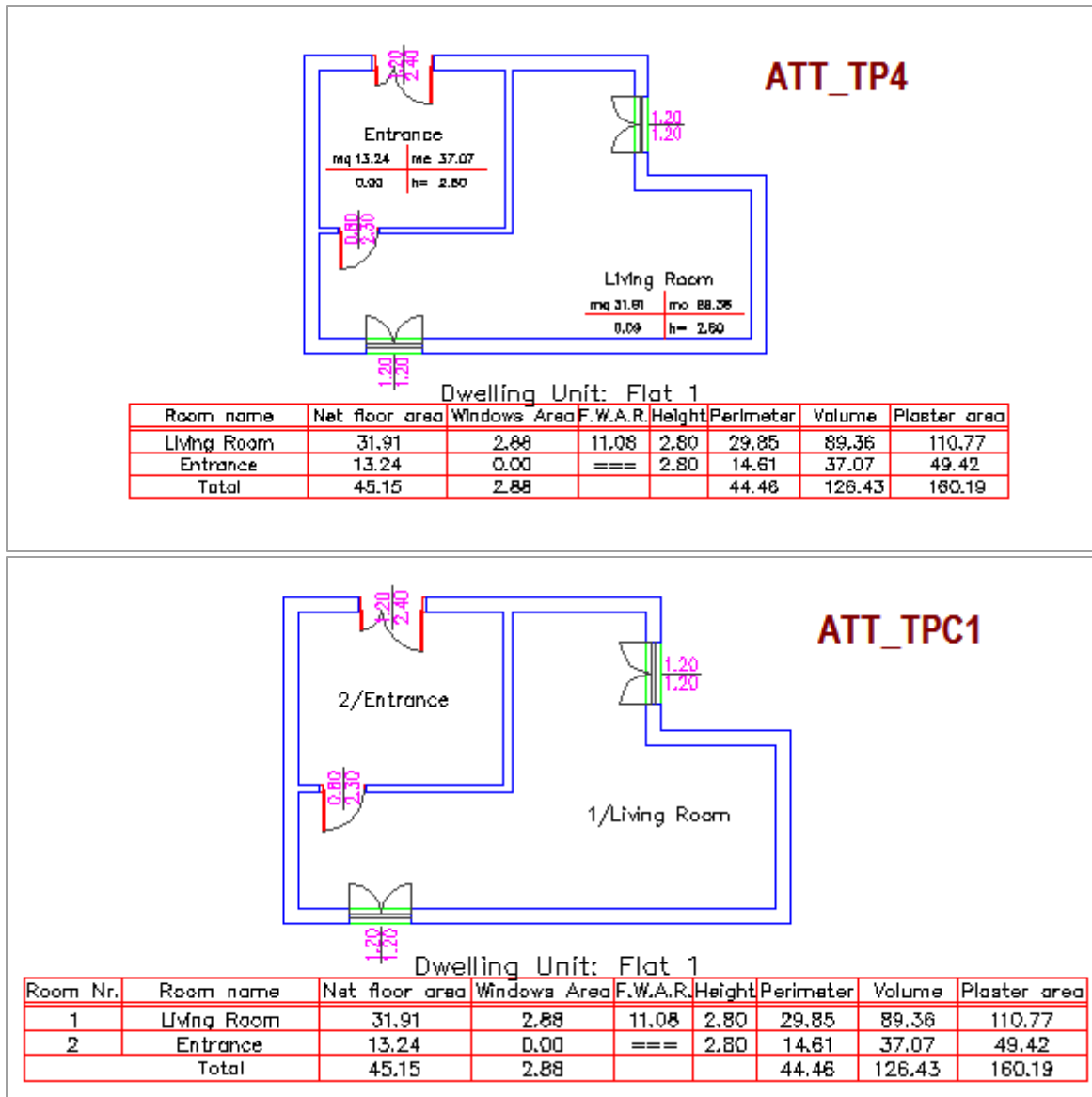
Room Nr.	Room name	Net floor area	Windows Area	W.A./N.F.A.
1	Living Room	31.91	2.88	0.09
2	Entrance	13.24	0.00	===
Total		45.15	2.88	

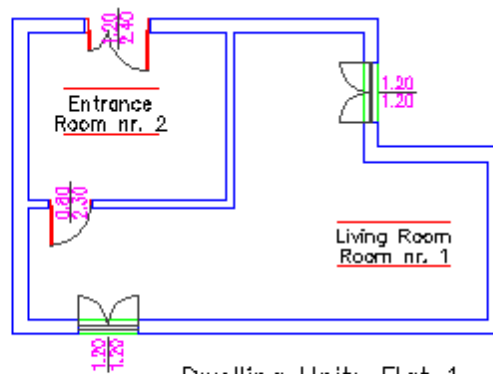


ATT_TP3

Dwelling Unit: Flat 1

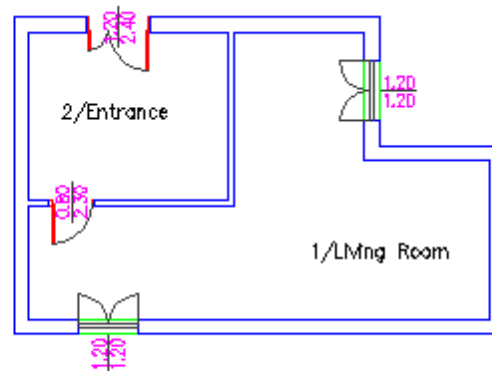
Room Nr.	Room name	Net floor area	Windows Area	F.W.A.R.
1	Living Room	31.91	2.88	11.08
2	Entrance	13.24	0.00	===
Total		45.15	2.88	



**ATT_TPC2**

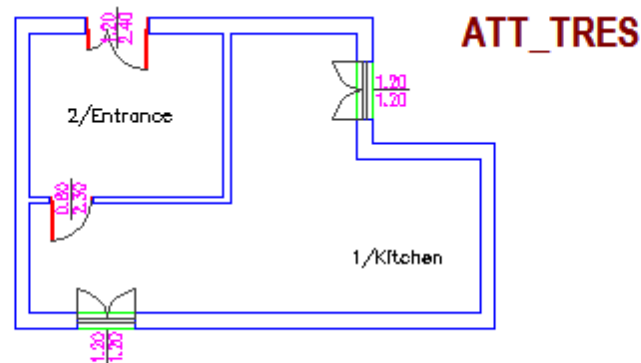
Dwelling Unit: Flat 1

Room Nr.	Room name	Net floor area	Windows Area	F.W.A.R.	Height	Perimeter	Volume	Plaster area
1	Living Room	31.91	2.66	11.08	2.80	29.85	89.36	110.77
2	Entrance	13.24	0.00	===	2.80	14.61	37.07	48.42
Total		45.15	2.66			44.46	126.43	160.19

**ATT_TPC3**

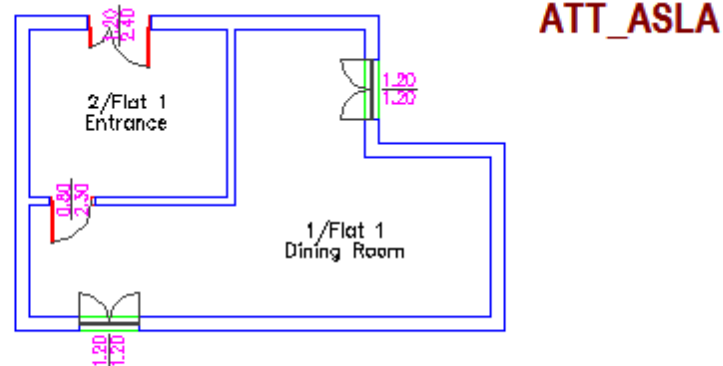
Dwelling Unit: Flat 1

Room Nr.	Room name	Net floor area	Windows Area	F.W.A.R.
1	Living Room	31.91	2.66	11.08
2	Entrance	13.24	0.00	===
Total		45.15	2.66	



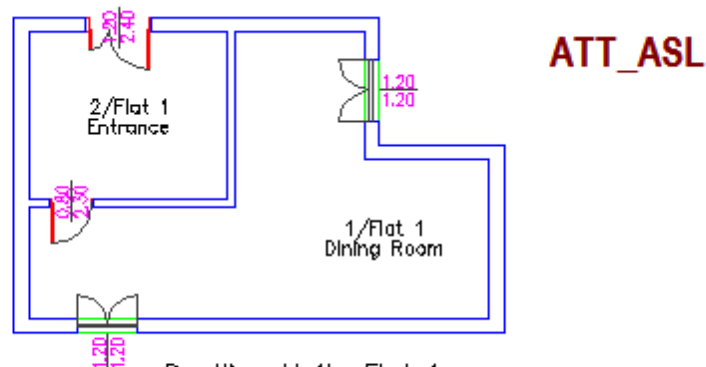
Dwelling Unit: Flat 1

Room Nr.	Room name	Living Space	Non-Living	Windows Area	W.F.A.R.
1	Kitchen	31.91		2.88	11.08
2	Entrance	13.24			
Total		45.15	0.00		



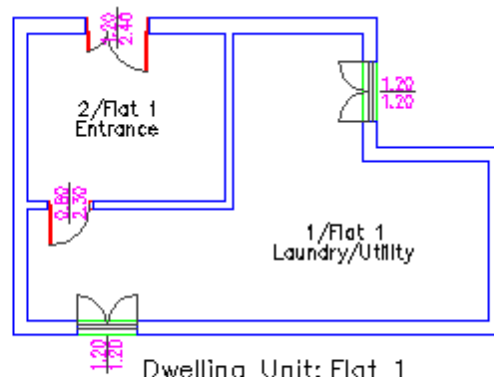
Dwelling Unit: Flat 1

Room Nr.	Room name	Net floor area	Windows Area	W.F.A.R.	Ventilation area	V.F.A.R.	Volume	Height
1	Dining Room	31.91	2.88	11.08	2.88	11.08	89.36	2.80
2	Entrance	13.24	0.00	0.00	1.66	7.88	37.07	2.80
Total		45.15	2.88		44.46		126.43	



Dwelling Unit: Flat 1

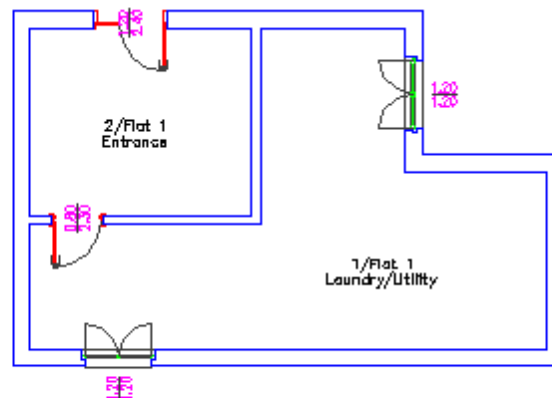
Room Nr.	Room name	Net floor area	Windows Area	F.A./8	Height	Perimeter	Volume	Plaster area
1	Dining Room	31.91	2.88	3.99	2.80	29.85	89.36	110.77
2	Entrance	13.24	0.00	1.66	2.80	14.61	37.07	49.42
Total		45.15	2.88			44.46	126.43	160.19



ATT_TRES1

Dwelling Unit: Flat 1

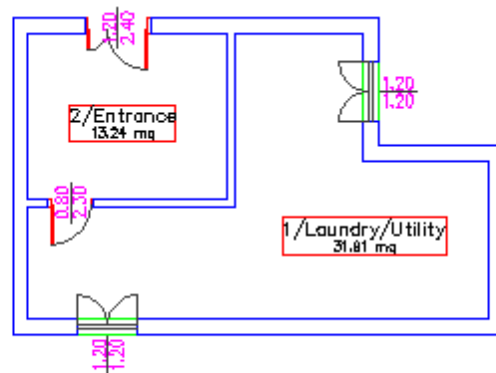
Room Nr.	Room name	Living Space	Non-Living	60% Non-Living	Windows Area	W.F.A.R.
1	Laundry/Utility	31.91			2.88	11.08
2	Entrance		13.24	7.94		
Total		31.91	13.24	7.94		
SUM OF LS + 60% NLS			39.86			



ATT_TRES2

Dwelling Unit: Flat 1

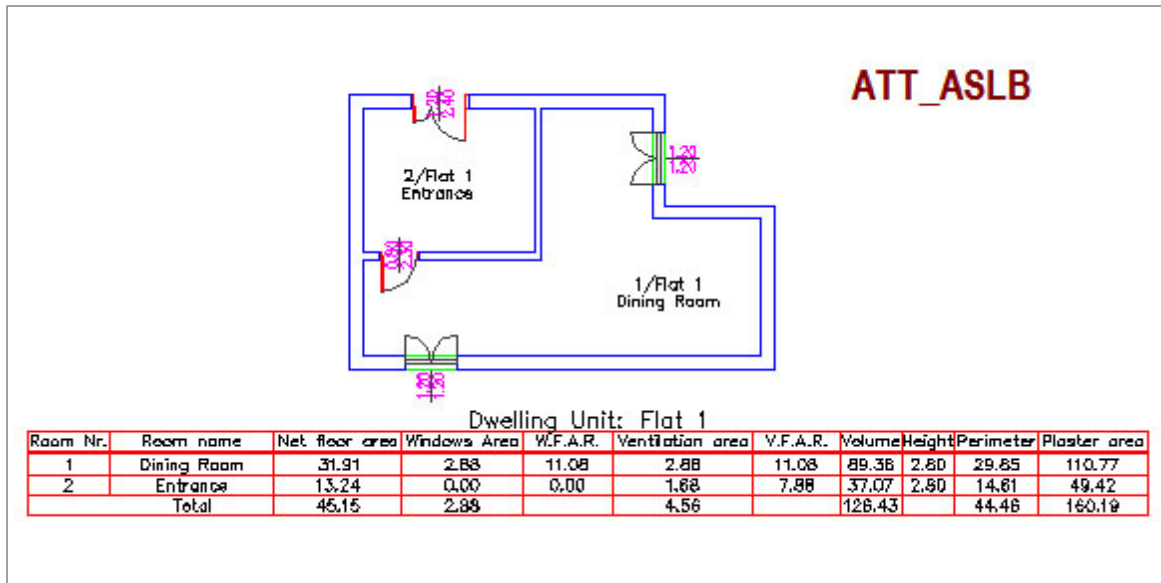
Room Nr.	Room name	Living Space	Non-Living	Windows Area	W.F.A.R.	Ventilation area	V.F.A.R.
1	Laundry/Utility	31.91		2.88	11.08	2.88	11.08
2	Entrance		13.24				
Total		31.91	13.24				



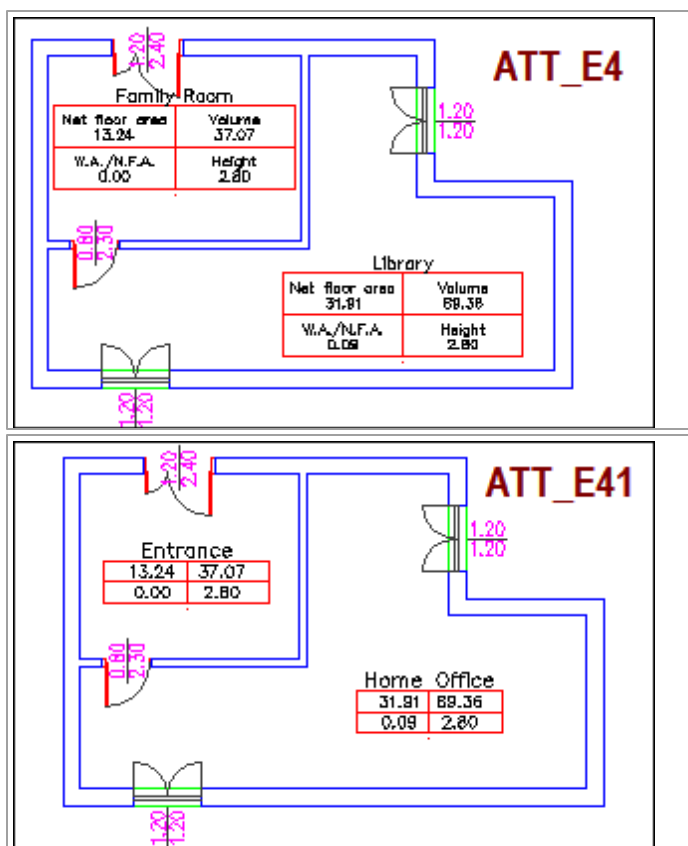
ATT_TREL

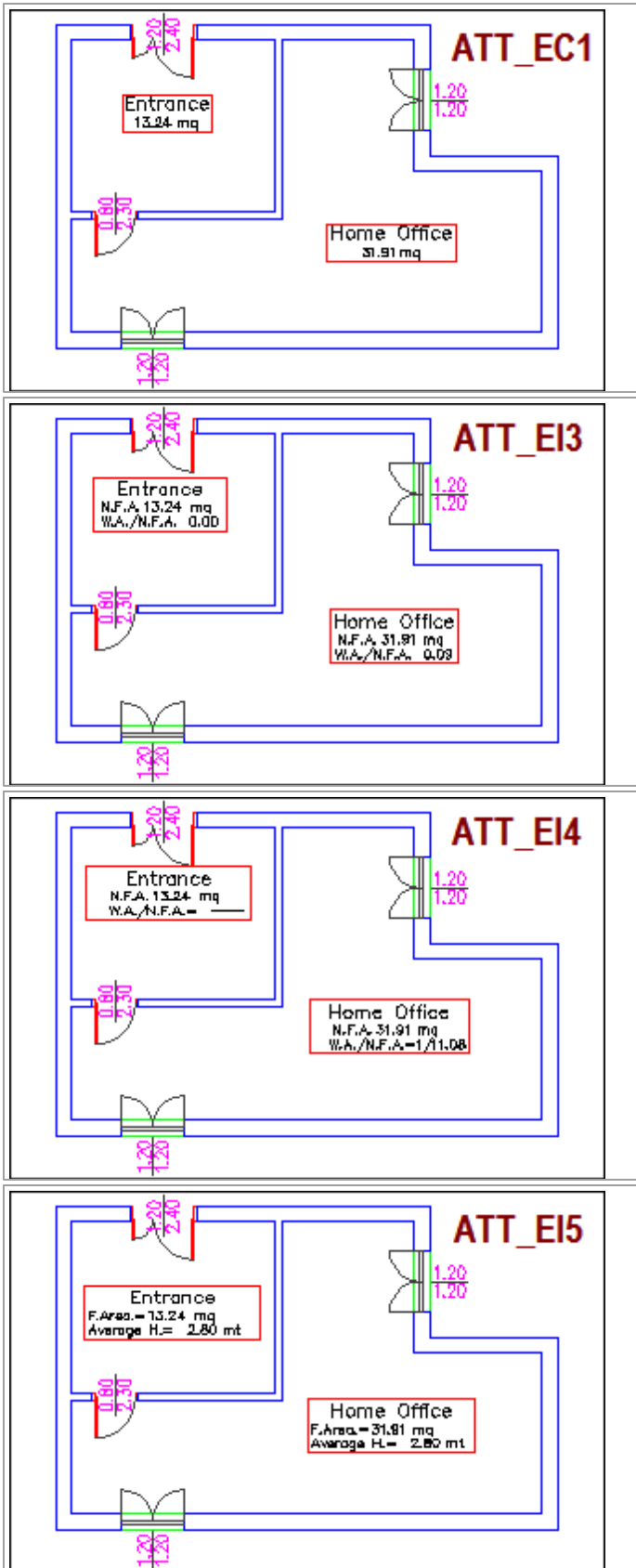
Dwelling Unit: Flat 1

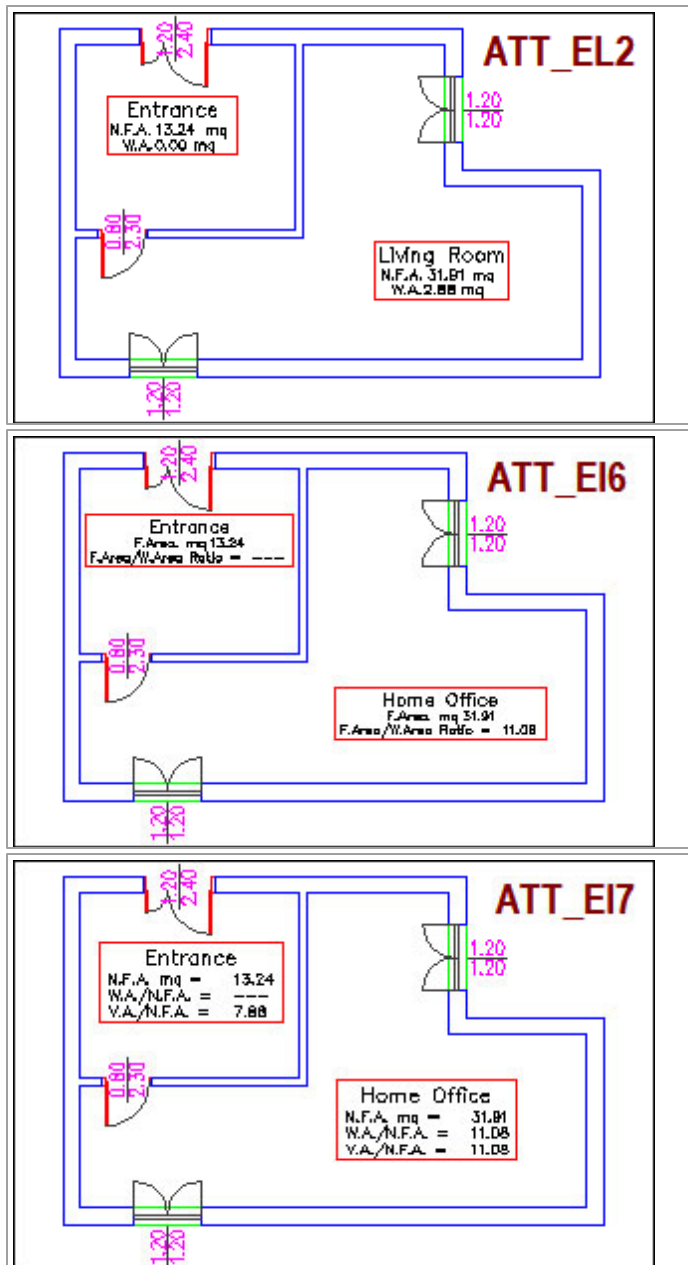
Room Nr.	Room name	Living Space	Non-Living
1	Laundry/Utility	31.91	
2	Entrance		13.24
Total		31.91	13.24



Room label without tab







Opening tabs

Opening indexing system

While openings are being inserted on walls, you will notice the presence of a [opening code](#) near the opening dimension. The frame code is composed of a prefix assigned as a parameter in the parameterization window and a suffix which is a whole number which automatically increases after each new opening is inserted. AddCAD manages two counters, one for windows and the other for doors and openings in general. The current value of the two counters can be set at any time. Obviously the new frames inserted will

start from the set code. for the counters, there are two commands to set them.

Command: DABC

Start number for door codes<1>:

DABC for the door counter and WABC for the window counter.

Command: WABC

Start number for window codes<1>:

Once the openings with a certain code have been inserted, you could wish to assign some of them with a different numerical code or even to re-index part of the layout. You can use the REINDEXABC command for this.

The command has the following messages:

Command: REINDEXABC

With which number do you want to start changing the code <1>:

Select the opening to which you want assign the code/Enter to finish:

Select the opening to which you want assign the code/Enter to finish:

..... and so on....

Inserting and updating opening tabs

There is one command called ABC which allows you to generate and recalculate opening tabs. The behavior of this command depends on the answer to the first message.

Insert object:abc_tab3

Opening code	Opening area	Width	Height	Room Number	Room Inside	Sup. Ventilation	Out Room Number	Room Outside

Group

Report of groups: Doors

Number of rows

Description	Current	New
Number of rows:	2	2
Group (Doors/Windows/Openings):	Openings	Openings

Parameter set: ☐ Show only important parameters

Last insert parameters

Last insert parameters

Windows

Doors

Openings

Cancel

Command: ABC

Select the opening table you want recalculate/Enter to insert a new one:

New opening tab

If you respond by pressing *Enter*, the parameterization window of the abacus tab will appear immediately. The window where the tab is inserted is the usual parameterization window. You will notice the presence of all three tab *parameter sets* used with AddCAD.

A tab for each floor and each opening category

For each floor, you can insert a tab for doors, one for windows and one for openings without frames.

Updating tab

If you select a tab, it is updated immediately. This means that all the data of the frames are recalculated and therefore the tab is modified.

Opening tabs and room calculation

The models supplied with the program have a great deal of information concerning the frames

and the rooms where the frames are placed. In order to write all the data in the tabs you must have calculated the room data before inserting or updating the tab.

The examples in the figures show that the three tab *parameter sets* are slightly different one from another. In particular the ADDCOA03 model visualizes the value of the *ventilation surface* for each frame, which is why it is important to have first completed a tab extract to check this value.

The figures reproduce the three models ADDCOA01, ADDCOA02 and ADDCOA03 currently available with the program. You can choose which of the two to use by writing the model name in the [Quantities survey system tab](#).

Tabs total

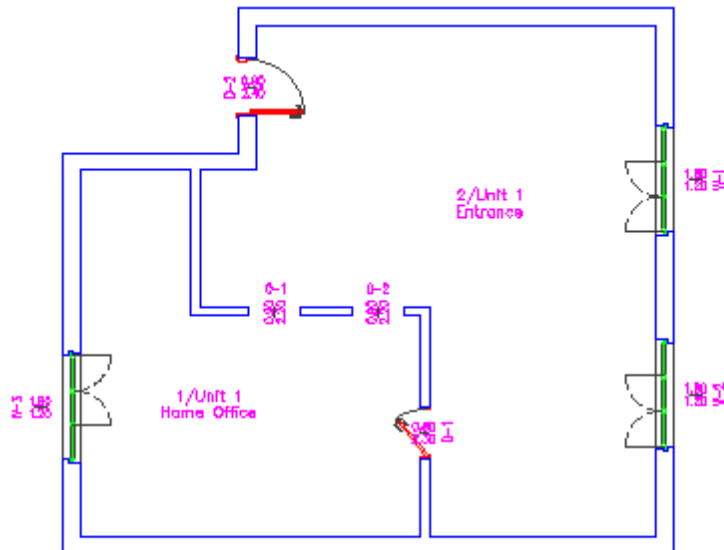
The command for calculating the totals and writing them in the tabs totals row is TOTABC, which only requires selecting the tab related to the total.

Height and style of texts

The text height of the labels and tabs is controlled by the [SCALEF command](#). To generate tabs, AddCAD uses two text styles, one for the heading row and one for the rows of data and for the labels. The names of these styles must be written in the [Parametric and stairs layer Tab](#) of AddCAD options. The definition of these styles can be changed at will.

Frame data extraction

Using the XABC command allows you to extract frame tab data in order to process it with other programs. The record path is described in a path description file called ADDCABC.DEX. This file has the same structure as the room data extraction file.



Openings tab model

ADDCOA01

Report of group: Windows

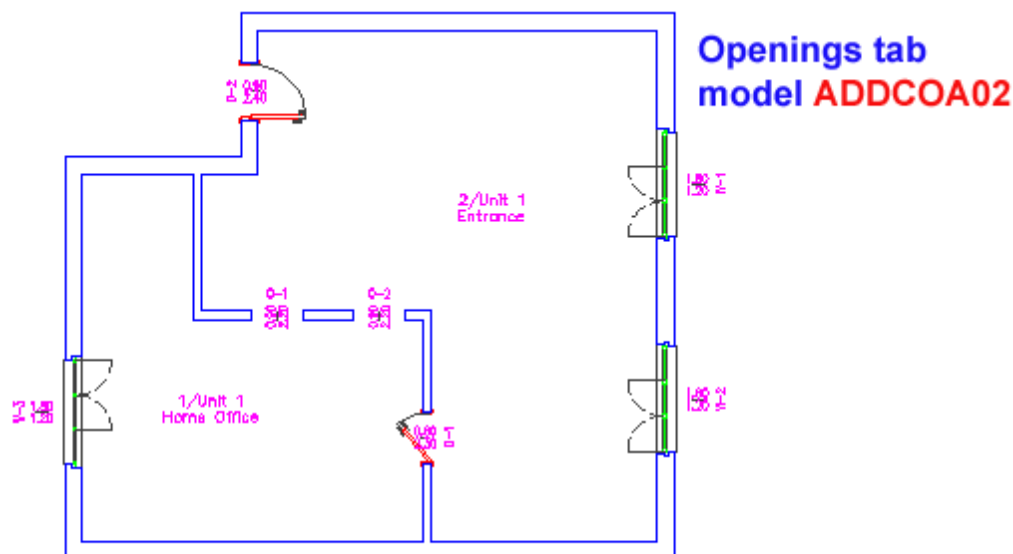
Opening code	Definition file	Material	Width	Height	In Room Number	Room Inside	Symbol
W-1	f3a	Wood	1.80	1.20	2	Entrance	W1
W-2	f3a	Wood	1.80	1.20	2	Entrance	W2
W-3	f3a	Wood	1.80	1.20	1	Home Office	W3

Report of group: Doors

Opening code	Definition file	Material	Width	Height	In Room Number	Room Inside	Symbol	Out Room Number	Room Outside
D-1	pa	Wood	0.80	2.30	1	Home Office	D1	2	Entrance
D-2	portes	Wood	0.80	2.40	2	Entrance	D2	0	

Report of group: Openings

Opening code	Definition file	Width	Height	In Room Number	Room Inside	Symbol	Out Room Number	Room Outside
O-1	arcal	0.90	2.20	1	Home Office	O1	2	Entrance
O-2	arcal	0.80	2.20	1	Home Office	O2	2	Entrance



Report of group: Windows

Opening code	Opening area	Windows area	Width	Height	In Room Number	Room Inside	Symbol	Bottom deduction	Top deduction
W-1	2.16	2.16	1.80	1.20	2	Entrance	III	0.00	0.00
W-2	2.16	2.16	1.80	1.20	2	Entrance	III	0.00	0.00
W-3	2.16	2.16	1.80	1.20	1	Home Office	III	0.00	0.00

Report of group: Doors

Opening code	Opening area	Width	Height	In Room Number	Room Inside	Symbol	Out Room Number	Room Outside
D-1	1.84	0.80	2.30	1	Home Office	II	2	Entrance
D-2	2.16	0.90	2.40	2	Entrance	II	0	

Report of group: Openings

Opening code	Opening area	Width	Height	In Room Number	Room Inside	Symbol	Out Room Number	Room Outside
O-1	1.89	0.90	2.20	1	Home Office	I	2	Entrance
O-2	1.89	0.90	2.20	1	Home Office	I	2	Entrance

Openings tab model ADDCOA3

Report of group: Windows

Opening code	Opening area	Sup. Illuminants	Width	Height	In Room Number	Room Inside	Sup. Ventilants	Bottom deduction	Top deduction
W-1	2.16	2.16	1.80	1.20	2	Entrance	2.16	0.00	0.00
W-2	2.16	2.16	1.80	1.20	2	Entrance	2.16	0.00	0.00
W-3	2.16	2.16	1.80	1.20	1	Home Office	2.16	0.00	0.00

Report of group: Doors

Opening code	Opening area	Width	Height	In Room Number	Room Inside	Sup. Ventilants	Out Room Number	Room Outside
D-1	1.84	0.80	2.30	1	Home Office	0.00	2	Entrance
D-2	2.16	0.90	2.40	2	Entrance	2.16	0	

Report of group: Openings

Opening code	Opening area	Width	Height	In Room Number	Room Inside	Sup. Ventilants	Out Room Number	Room Outside
O-1	1.89	0.90	2.20	1	Home Office	0.00	2	Entrance
O-2	1.89	0.90	2.20	1	Home Office	0.00	2	Entrance

Definition of new models

Just like for rooms, tabs can be defined with a format at will. The program has three standard models: ADDCOA01, ADDCOA02 and ADDCOA03. Each model includes three parameter sets: *Windows*, *Doors* and *Generic openings*. An opening tab can be created for each of these parameter sets and for each floor.

AddCAD license and version

The WHO command lets you identify the program by displaying copyright information. It shows the information on the license and the version of AddCAD.



The command is activated from the *AddCAD Base ->About AddCAD....* drop-down menu or from the button on the toolbar as show in the figure. Clicking on the button the following window appears showing the information about the license and the current version

of your program:



The window shows the activation type you choose when you initially activated AddCAD: USB/Computer or *Demo* if you are still running the trial version. This information is quite useful in case the program remains unused for a long time and you might want to check from [AddCAD website](#) if newer upgrades or versions are available.

Block catalog

User block library

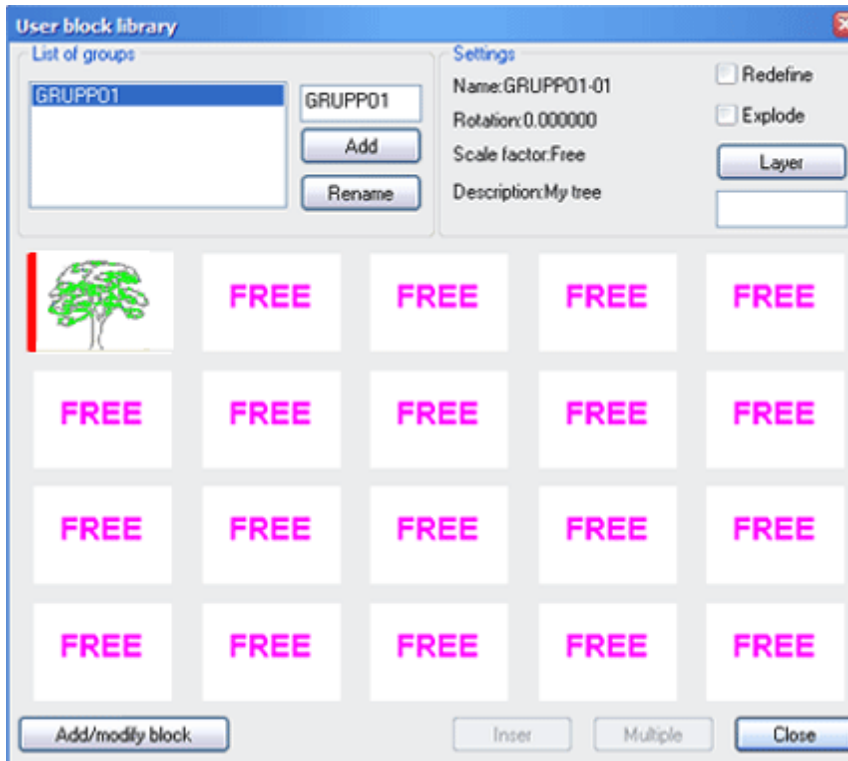
AddCAD provides the CATBLOCKS command which allows you to manage a block library created by the user. This system memorizes and inserts blocks created with

AutoCAD. Being easy-to-use and being able to use [insertion tools for floors](#) makes this command extremely useful.

Information concerning block catalog and security copies

It is important to remember that AddCAD memorizes all information concerning blocks, drawings and images in the CAT folder which must always be in AddCAD's main directory.

Another important file is called *catalog.ini*, again in the CAT folder. When you make library backup copies, just save a copy of the CAT folder in a safe place.



The CATBLOCKS command fundamentally works with two dialog boxes.

The first is used to search for and insert symbols or blocks, while the second defines or adds new blocks and symbols and modifies existing ones. The library is composed of block groups, each of which can contain up to twenty blocks. You can define as many groups as you wish.

Managing groups

Groups are managed at the top left of the dialog box. You can select a group from the list, change its name or create a new group.

Features of selected block

The block selected is described in the area above to the right. During the definition stage, certain characteristics can be assigned to each block. It is possible to specify special characteristics during this stage and only for this insertion. We can decide whether to explode the block after insertion, whether to redefine a block present in the drawing and the layer name where the block is to be inserted.

The block features visualized in the viewport are the description of the selected block, the name of the selected block, its scale factor and the rotation to be applied to the selected block.

Images of blocks of group

The four rows of images visualize the blocks of the selected group. Each position corresponds to one block. If a position is free, an image indicating *FREE* appears. If a position is taken up by a block, but the corresponding image is not present in the CAT directory, probably because it was removed or was not created, then the text *Cannot find the preview of the block* appears. A block can be selected by simply moving your mouse

on an image and left-clicking. The selected block will have a red line on the left. AddCAD memorizes the selected position to reuse the command in the future and to allow repeat insertions.

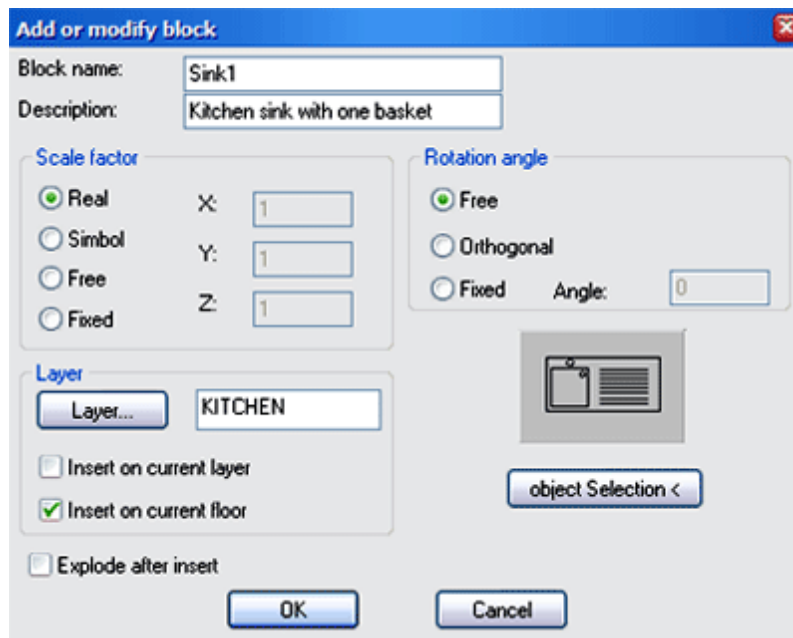
There are four buttons at the bottom of the dialog box. *Close* obviously exits the command without modifying the drawing or block catalog. The *Insert* and *Multiple* buttons are used to insert a block in the drawing. The first button inserts it once and the other inserts it repeatedly, until the user presses Enter.

Add and modify a block definition

The *Add/Modify block* button is used to define a new block or to change the definition of the selected block. You must therefore be careful to select an image indicating *Free* to be able to add a new block to the library. If you select an image which already has a block, its definition is modified. After having chosen *Add/Modify block*, the [dialog box is shown](#) which allows you to define a block in the selected position inside the main dialog box.

Add and modify a block definition

The definition of a block consists in selecting some objects, its point of insertion and attributing the block with a series of characteristics.



Block name and Description

It is possible to assign a block with a name. Keep in mind that the block name is also the DWG file name of the block and of the BMP image. Both are memorized in the CAT folder on the hard disk. Its description can be added to the block so it is easier to remember the type of object represented by the block.

Object selection and insertion point

The button *object Selection <* allows you to select the entities of the new block. In fact the window closes temporarily and

the following message is displayed:

Select the objects of the block:

Insertion point of the block:

End selection of the entities by pressing Enter.

Creating BMP image

To create a BMP image, we recommend the plot with the AutoCAD Driver PublishToWeb JPG.pc3. A customized 80x65 Pixel sheet size must be set.

The best way to plot is by using the window option, selecting the area containing the block and plotting with *Adapt to sheet*. The result is a JPG image which when opened with Paint can be saved in BMP.

Scale factor

Real

The block will be inserted as a real element. This means that the dimensions of the block will be adapted to the unit of measurement being used (meters or centimeters), but they remain exactly as in the definition. We recommend defining the block by working in centimeters to maintain coherence with AddCAD.

Symbol

The block will be inserted as a symbol. This allows you to insert the block by following the chosen scale factor with the [SCALEF command](#). Again the block should be drawn working in centimeters. Furthermore it is recommended to draw it so that it is the size you want when plotted at 100%.

Free

The scale factor will be requested when the block is inserted in the drawing, just like when inserting a block with the INSERT command.

Fixed

The block will be inserted with the scale factor indicated in the X: Y: Z: boxes. Therefore you must write an insertion scale factor.

Rotation angle

Free

The rotation will be requested when inserting the block.

Fixed

The block will be inserted with the indicated rotation. Rotation must be expressed in degrees.

Orthogonal

Rotation will be requested during insertion of the block in the drawing and the orthogonal mode will be enabled.

Layer

If *Insert on current layer* is enabled, the block will always be inserted in the current layer without any link with the floor elevation. If *Insert on current layer* is not enabled, the layer must be assigned a name. It is possible to insert blocks in the [current floor](#) of the drawing, both from a viewpoint of the insertion dimension and from a layer name viewpoint. If *Insert on current floor is enabled*, the layer name will be joined with the current floor elevation. You can either write the layer name or select it from one of the layers in the drawing.

Explode after insert

This establishes whether the block must be inserted exploded or not.

Layer utility

Empty all entities from the layer

This command, LAYERDEL, is very useful as it deletes all the entities belonging to a certain layer. The layer itself will not be eliminated. If you wish to do so, you must use the AutoCAD PURGE LAYER command. The command simply requests the layer to be empty.

Command: LAYERDEL

Specify the name of the layer to empty or Enter to select:

If you press *Enter*, the command continues by asking:

Select an object from which to copy the layer name:

If you select an entity, first the command finds the layer which that entity belongs to and then deletes all the entities of the layer. This is a very convenient operation but can be destructive. Fortunately the UNDO command of AutoCAD can correct any errors caused by unintentional use of this command.

Layer activation commands

The LAYEROFF command allows you to disable a layer by selecting an entity. The layer name is copied from the entity.

LAYERFRZ freezes a layer after an entity has been selected.

The commands therefore work the same way.

Command: LAYEROFF

Select an object on the layer to turn off:

Command: LAYERFRZ

Select an object on the layer to freeze:

The layer can be indicated by selecting an entity belonging to a block. Any attempt to freeze the current layer ends up with an error message.

Visualize only selected layers

The LAYERISO command allows you to select entities whose layers are left active, while all other layers are disabled. The result is that of isolating the selected layers. The command also works with entities belonging to blocks and can therefore be used with parametric objects and other AddCAD objects.

Command: LAYERISO

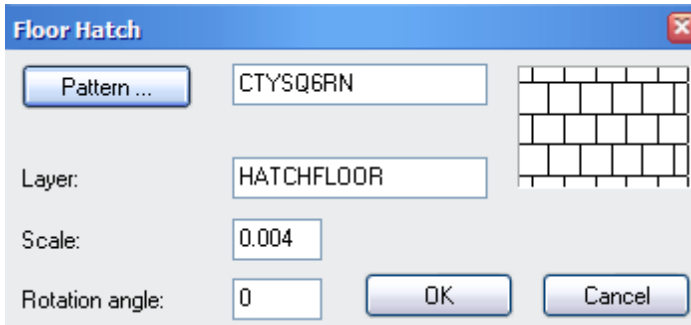
Select an objects on layer to leave visible, Enter to exit:

Select an objects on layer to leave visible, Enter to exit:

....

Floor coloring and dithering

The HFLOOR command is used to hatch or color rooms. This command also considers the [functional lines](#) in the drawing, making it possible to color the various functional areas inside large rooms differently. Furthermore AddCAD parametric objects, such as stairs and furniture, contain encumbrance information which form islands inside the hatches or colors of the rooms. One limit of these areas is that of not managing intersections or overhangs outside of the rooms. If therefore you abide by the rule of not overlapping objects and fitting them inside the room, the areas will be colored correctly.



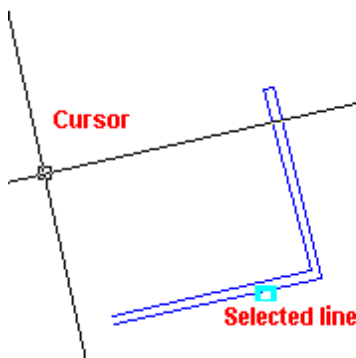
The dialog box is used to acquire data for the hatch. The *Pattern...* button chooses the desired model. Remember that AutoCAD includes a model called SOLID to color an area. When finished, the command requests a point to identify the room.

Command: HFLOOR

Pick internal point of a room or Enter to exit:

The HFLOOR command performs a path search such as calculation of room data. The conditions for correct operation are therefore the same as [room calculation](#).

Cursor rotation



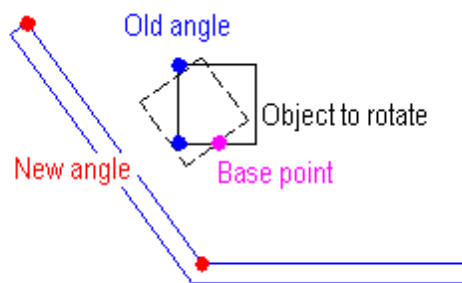
The ROTSNAP command rotates the axes based on the selected entity. This is a very useful extension of AutoCAD's SNAP command which is often a valid alternative to UCS rotation. It allows you to draw the selected line orthogonally.

Command: ROTSNAP

Select the reference line or Enter to set to zero:

Select the entity according to which you want to rotate the cursor or press Enter to restore that parallel with the coordinate axes.

Rotate with a second reference



The ROTATEAS command allows you to rotate a set of entities indicating the new angle by means of another entity completely detached from the angle of reference. For some rotations, AutoCAD's ROTATE command has a few limits. It is not possible to indicate the start and end angle with four points. Consider the following example.

Command: ROTATEAS

Select entities:

Base point:

Specify the old reference angle:

Second point:

New angle:

Second point:

The last two requests make it possible to assign the angle by selecting points on another entity.

Move in Z direction

Move in Z axis direction

MOVEZ displaces the selected entities in the Z direction. In some cases, it can remind you of the AutoCAD CHANGE PROP ELEV (change elevation) command. Actually, whatever is the current UCS, the MOVEZ command always displaces the selected entities in the Z direction of the global UCS. Furthermore the MOVEZ command requests the numerical displacement value. If the number is positive, then movement will be in the positive Z direction; if the number is negative, then movement will be in the negative Z direction.

Command: MOVEZ

Amount of the displacement:

Select entities:

Assign Z elevation

The ASSIGNZ command is used to move an entity by assigning a certain elevation value. Unlike MOVEZ, which moves entities along the Z axis, ASSIGNZ assigns the entity with the required Z value. The requests are the following.

Command:ASSIGNZ

Z elevation where you want put the entities:

Select entities:

Offset with layer change

The OFFSETCL command works exactly like AutoCAD's OFFSET command except for a very useful detail during the drawing stage. It allows you to simultaneously change layers on the new object created for displacement.

The dialog is the following.

Command: OFFSETCL

Specify the offset distance<0.10>:

Select object to offset:

Specify point on side to offset:

.....

The command repeats the last two questions until the user presses Enter. The displaced entities are placed on the current layer, whatever layer the selected object has. Remember to make the desired layer current before launching the command.

The current version of the OFFSETCL command only processes lines, arcs and circles. It does not consider other entities.

Line rectangles

The RECTANGL command allows you to immediately draw a rectangle with sides parallel to the coordinate axes. The drawn entities are lines. The x and y axes are always those of the global UCS. In fact the requests regard the endpoints of one of the diagonals. One special feature of this command is at the lines are placed on the current wall layer. The program therefore considers the current floor elevation and the layer currently set for walls and uses the corresponding layer to generate lines. This makes it especially useful when drawing rectangular rooms or quadrilaterals in general.

Comando: RECTANGL

Specify the first endpoint of the diagonal:

Specify the second endpoint of the diagonal:

Breaking entities

Command BREAKA

The BREAKA command allows you to divide an entity at one point without deleting parts of the same entity. It corresponds to AutoCAD's BREAK command with the option F(irst) followed by the character "@" to confirm the first point indicated as the second breakpoint.

Command: BREAKA

Select the object to break:

Specify the breakpoint:

Command: BREAKX

The BREAKX command allows you to break an entity in two points, deleting the part in between.

Command: BREAKX

Select the object to break:

Specify the first breakpoint:

Specify the second breakpoint:

All of the entities which can be used with AutoCAD's BREAK command can be selected with BREAKA and BREAKX.

Properties of entities

These commands allow you to change the color, linetype and layer of the selected entities. The capability of selecting another entity to identify the new property allows them to be used immediately with respect to the AutoCAD property management.

Change color

Command: CHANGE C

Select object to change color:

Select an object from which to copy the color or Enter to select a color:

It is possible to select an object from which the color can be copied or to answer pressing Enter. In this case the AutoCAD color dialog box is shown in which you may pick the color to assign the selected entities.

Change layer

Comando: CHANGEL

Select object to change layer:

Specify layer name or Enter to select an object from which to copy the layer name:

<ENTER>

Select an object from which to copy the layer name:

If you answer the second question with Enter, the command continues by asking the entity from which the layer name must be copied. If you answer with a non-existent layer name, the command automatically creates a new layer.

Change linetype

Command: CHANGELT

Select object to change linetype:

Specify linetype name or Enter to select an object from which to copy the linetype:

<ENTER>

Select an object from which to copy the linetype:

You will notice that the linetype has no meaning for some kinds of entities. If a non-existent linetype is indicated in the drawing, it is uploaded automatically from the AutoCAD support files.

Change text attributes

This command is capable of changing the height and style of all the selected texts. The command has two options.

Command: CHANGET

Scale factor to apply to the text objects [Style/Height]:

If you answer with a number, it will be intended as the scale factor to be applied to the texts. If the scale factor is higher than 1, all the selected texts will be proportionally enlarged; if it is less than 1, they will be reduced.

If the *Height* option is chosen, the command allows you to change height.

The *style* must already be defined in the drawing. The points of insertion and other features of the texts do not undergo any modification. Texts can be selected by using any selection method. If the selection includes entities not linked to the text, the command will ignore them.

Multiple distances

AutoCAD's DIST command allows you to calculate the distance between two points in the drawing. This command has the setback of always needing to insert both points even

when you wish to repeat the command. The ADISTANCE command is an advanced command for calculating distances.

The command has the following options.

Comando:ADISTANCE

Kind of measurement [Origin/Multiple]<Origin>:

Specify first point:

Specify next point:

Distance:3.3283 dx:3.1609 dy:1.0424 dz:0.0000

Specify next point:

....

The command ends by pressing *Enter* when a second point is requested.

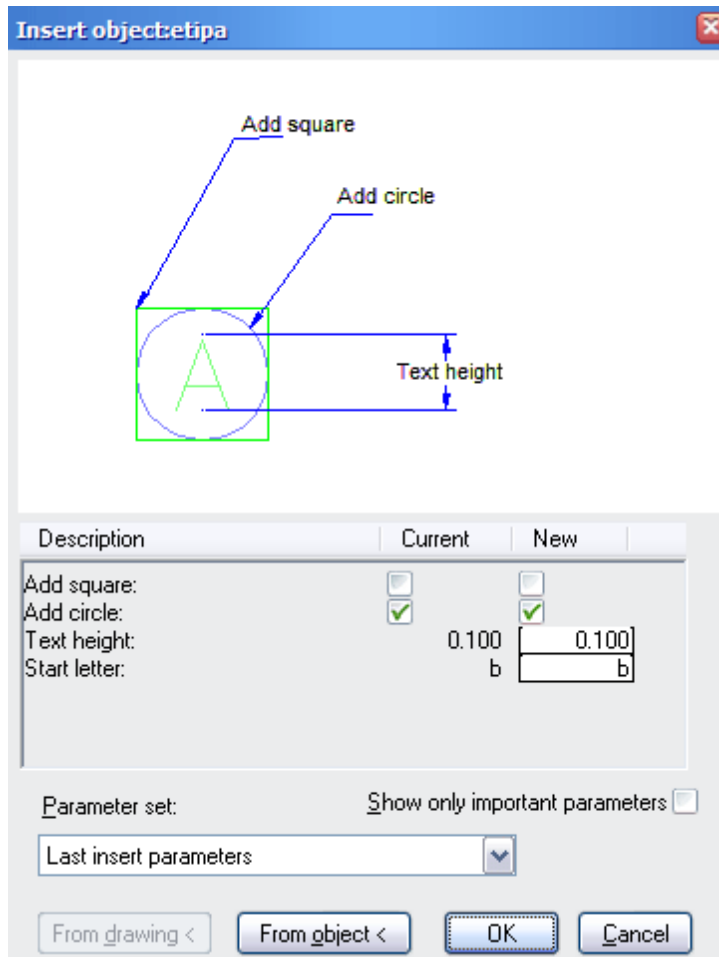
The *Origin* option sets the first point for all the following distances. The *multiple* option uses the second point of the previous distance to become the first point of the next distance.

Aside from showing the partial distances of each point (also divided in x, y and z), the command carries the sum of the partial distances at the end of the command.

Numbered labels



Numbered labels are graphical elements frequently used in detail drawings. AddCAD makes both letter and number labels available. There is just one command allowing you to modify the labels by reassigning the number or letter progressively.



The AddCAD labels are special parametric objects making it possible to choose some options in the object insertion window such as the presence of a circle or square around the number or letter. Once you have chosen the options, the program continues by requesting the points of insertion, progressively increasing the letter or number after each insertion. The REINDEXLABEL command allows you to change the numbering order of the labels. The command asks which number or letter you wish to start from and then asks you to select the labels to be corrected.

Command: REINDEXLABEL

Number or letter with which to start insertion of the progressive labels:1

Select a label that you want assign the number 1, Enter to exit:

Select a label that you want assign the number 2, Enter to exit:

....

If you start with a letter and labels with numbers are selected or vice versa, the result will not be as expected. It is not possible to transform number labels into letter labels and vice versa.

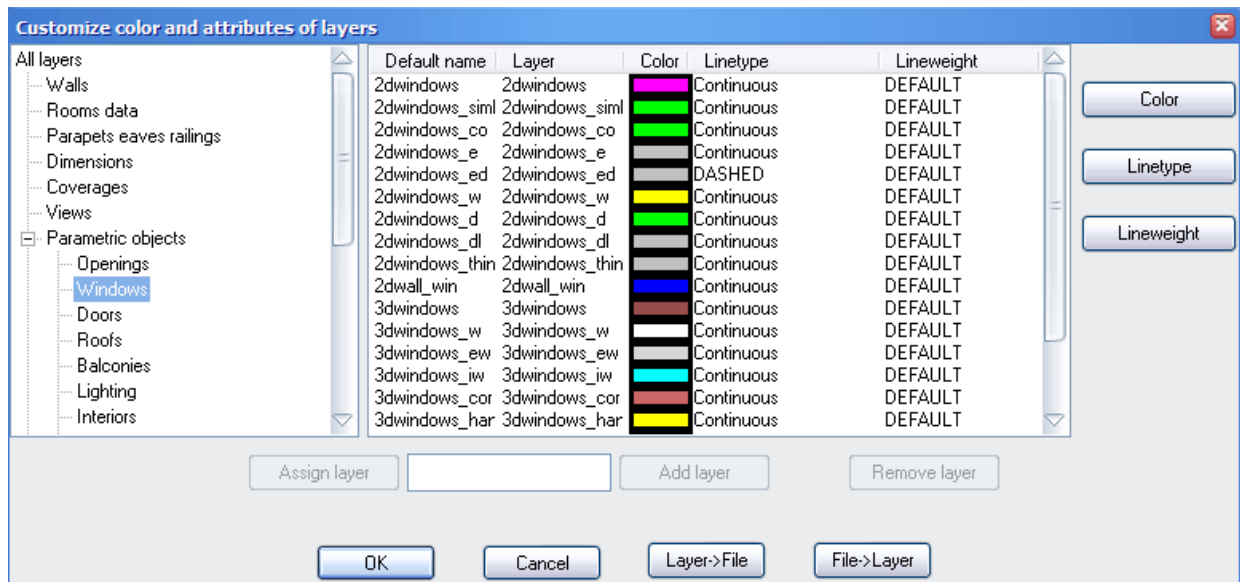
Layer generation attributes

When a parametric object is inserted, AddCAD places the graphical entities belonging to the object [on various layers](#). AddCAD automatically generates all the layers of the object. All the entities of the object are created with BYLAYER color. Using AutoCAD we know that in this case the color of the entity is the one associated to the layer it belongs to.

Therefore once the object has been generated, the color of the various entities can be changed by changing the color of relative layers.

When AddCAD generates new layers, both to insert new objects and to use the various commands, each is associated with a color, a linetype and linewidth, the values of which are taken from the *layerc.def* configuration files. Each row of this file describes all the generation attributes of the layers.

The file includes all the layers used by AddCAD's parametric objects library and commands .



If you wish to modify the attributes with which layers are generated, the file can be changed with a command which considerably simplifies this operation. The DDLCOLOR command opens a dialog box inside of which layer generation attributes can be quickly modified and a standard for layer names set.

List of layers

The list of layers allows you to make a multiple selection so that several layers can be changed simultaneously. The first column has all the layers which AddCAD can generate as a result of the insertion of objects and the use of commands. The layers are ordered by command and object groups, and inside the groups in alphabetical order.

OK

Modifications of the layer attributes are memorized in the configuration file. If you exit by pressing OK, the layers already in the drawing will not be modified, whereas the subsequent creations of new layers will have the new attributes.

Updating layers

The *File->layer* button is pressed to update the layers in the drawing. The layers already

present in the drawing take on the attributes of the list memorized in the file. This is very convenient if drawings are already made and you wish to quickly update the layers.

Updating files

The *Layer->file* button is pressed to update the *layerc.def* configuration file according to the attributes of the layers in the current drawing. This function is used to immediately modify the *layerc.def* file. First the color, linetype and lineweight attributes of the layers are set in the drawing and then they are all updated by pressing this button.

Generating layers with different name

When the program is installed, the second column is the same as the first. The second column allows the user to change the name of the layers actually created in the drawing. This makes it possible to create less layers to be used by AddCAD. To the right of the *Assign layer* button, you may write a name and assign it to all the layers selected.

Adding new layers and eliminating layers from the list

Layers can be added and deleted in the layer tab by means of the two buttons, *Add Layer* and *Remove layer*.

If you select a layer, it can be deleted.

If you write a new layer name in the box to the left of the *Layer* button, the new layer will be inserted above the row selected in the tab.

Color

The Color button allows you to choose a color and to assign it to selected layers.

Linetype

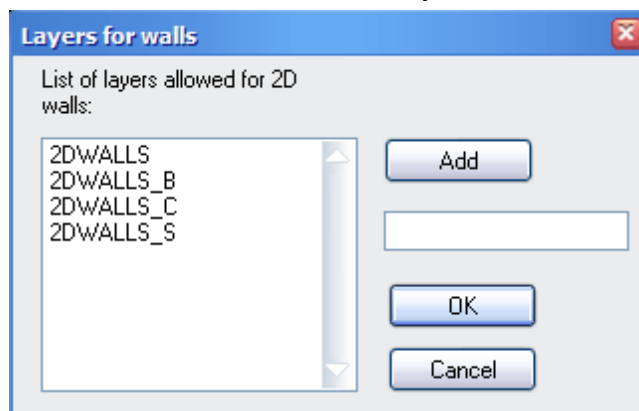
The Linetype button allows you to select a linetype from the AutoCAD lines definition file and to associate it to the selected layers.

Lineweight

The Lineweight button allows you to select a lineweight and to associate it to the selected layers.

Names for wall layers

The LWALL command allows you to define the list of variable parts of [wall layers](#).



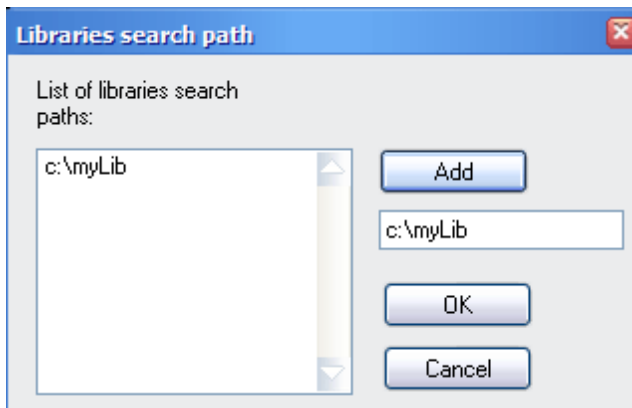
Just write a name in the edit box of the window shown in the figure and select *Add* so that AddCAD can use that name as a layer upon which wall lines and arcs are found.

Only the variable parts of the names are found in the list to the left. When AddCAD draws a wall on that layer, the layer name actually becomes `ADDC<name>$<FE>` where in place of `<name>` there will be one of the names of the list and in place of `<FE>` the *Floor Elevation* upon which you

are working. The walls defined in this list will be offered for selection when walls are drawn and stratifications defined.

Searching definition files

When AddCAD is installed, all the search paths of the program support files are configured, including that of the parametric objects files of the standard library supplied with the program. The LIBRARY command allows you to specify a list in the additional directory where AddCAD looks for the object definition files, the object image support files and the blocks inserted from the block menu. The paths specified with this command have the priority over standard paths. Therefore if there are two objects with the same name (one in the user library path and the other in the standard library), the first object will be uploaded.



The command allows you to add search paths through a dialog box.

A new library directory can be specified in the edit box. Once the name has been written, press *Add*. By selecting a library in the list, it can be deleted from the list by pressing *Delete* on the keyboard.

The data of the list is memorized in the current drawing. If you wish to permanently add the library paths and always find them in the new drawings, use the LIBRARY command in the starting drawing template

and then save the modifications.

AddCAD options

All the option data configured in the various dialog boxes and the default answers to the requests of the various commands are memorized in the current drawing as the commands are used. AddCAD option data and system variables are saved with the drawing when closed. The next time it is opened, the program will set the values used the last time as default values.

Drawing templates

AddCAD is supplied with various drawing models, *addcadmt.dwt*, *addcadcm.dwt* and *addcadmm.dwt*, which are set up respectively to work in meters, centimeters and millimeters. They have a whole series of defaults regarding system variables, metric quantities survey system and dimensions. Everyone can modify system values as needed and save them in a template from which to start the new drawings.

There is one way of bringing the system values to a base configuration. The *Parametric and stairs layer tab* of the AddCAD options dialog box has an edit box called *Drawing units*. If you modify this field and then exit pressing *Apply*, all the values are reconfigured respecting the indicated multiplication factor. It's needless to say that it is graphically confusing when a unit of measurement is changed in a drawing already drawn with AddCAD. Note, use this option with caution.

The ADDOPTIONS command allows you to change many settings of the drawing and command options. The dialog box has five tabs:

[Floors and views generation](#)

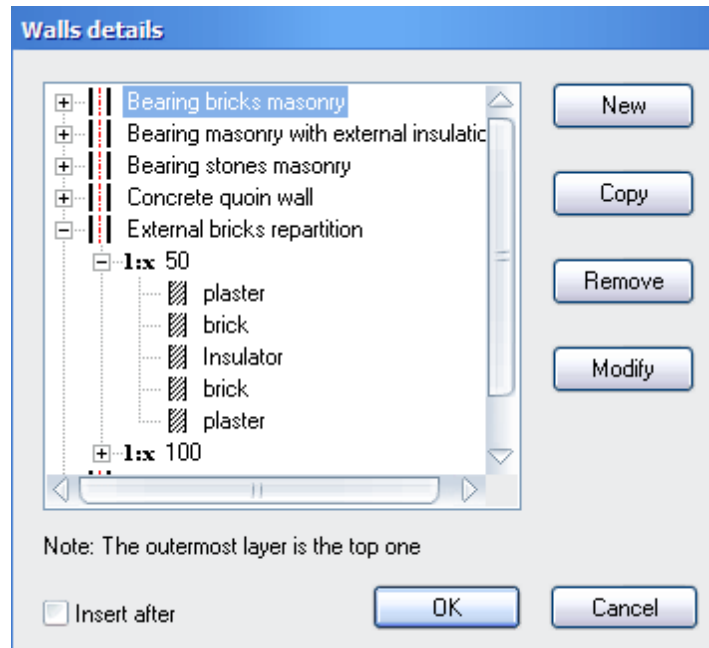
[Openings and walls 2D/3D](#)

[Dimensions](#)

[Parametric and stairs layer](#)

[Quantities survey system](#)

Definition of new stratification



The DEFMXT command is used to manage the definition of wall types. New types can be added and existing types modified. The definition of a wall type entails the insertion of some general information and the definition of various stratifications depending on various plot scale factors.

In general, wall details are drawn for small scale factors. Standard wall types provided with AddCAD have a detailed graphic representation only for the scale factor 1:50. The stratification of the 1:100 scale factor is one filled model for the entire wall. No stratification detail is used for larger scale factors.

If you launch the DEFMXT command,

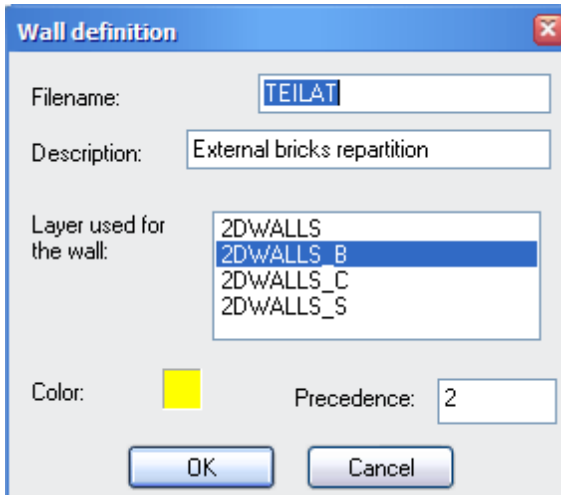
you can access the wall definition dialog box.

The tree structure allows you to access the data which is logically shown on a lower level by clicking the symbol “+”, or to narrow view by clicking “-”.

Stratification is defined on two levels. In the first level, we list the scale factors and in the second the material stratifications for the various scale factors, which are listed in order from the outermost to the innermost. We can see in the figure how for ‘*External bricks repartition*’ two scales have been created, 1:50 and 1:100. For the 1:50 scale, we have defined the stratification starting from the outside as Plaster, Brick, Insulation, Brick and Plaster, while for 1:100 only Wall, because as the scale factor increases, we want less details.

It can be noted that for correct processing of asymmetric stratifications, the outer stratification must be at the top. Stratifications are normally added on top of the one selected; to insert them below, enable the *Insert after* control button. Exiting the dialog box by pressing *OK* saves the modifications in the relative files, if you exit pressing *Cancel* no modification is made.

Wall features and the individual stratification can be modified by selecting the wall which we wish to modify and selecting *Modify*.



The following dialog box allows you to define the general features of the wall. It is opened by selecting the description of the wall to be modified. Pressing *New* allows you to define a new wall type.

The attributes have the following meaning.

Filename

The filename mxt is used to memorize the wall stratification being considered. You do not need to insert the mxt extension or the path, which will always be the root AddCAD path.

BMP wall type image

When you create a wall, you must also create a bitmap which graphically represents the wall in the dialog boxes regarding the choice of the

wall type. This BMP file will have the same name as the mxt definition file.

Description

This description will be found in the references made inside the program, for example in the wall drawing, metric quantities survey system, etc.

Layer used for the wall

The layer selected automatically by the WALL command when you choose to insert this type of wall. The list of walls can be defined with the [LWALL command](#).

Color

The color to recognize the wall when assignments are made with the [DEFWALL command](#).

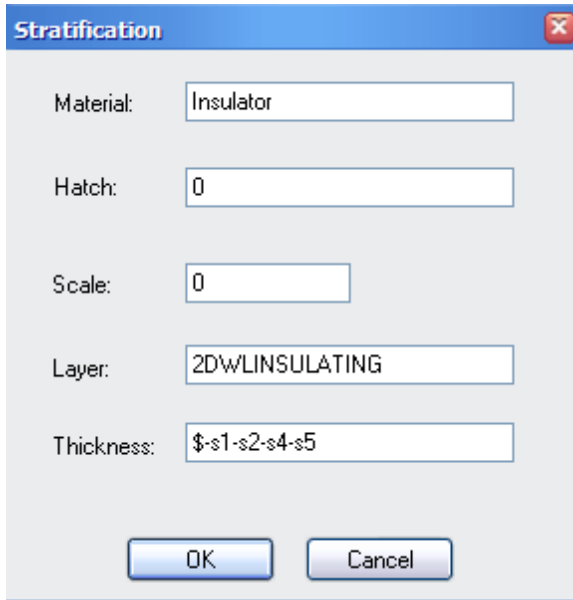
Precedence

The overall priority of the wall type.

Modifying or adding a stratification

If you select a stratification in the main window and you press *Modify* or *New*, the definition window of the individual stratification opens.

The attributes have the following meaning.

A screenshot of a software dialog box titled "Stratification". It contains five input fields: "Material" with the text "Insulator", "Hatch" with the value "0", "Scale" with the value "0", "Layer" with the text "2DWLINSULATING", and "Thickness" with the expression "\$-s1-s2-s4-s5". At the bottom are "OK" and "Cancel" buttons.**Material**

The description of the material which will appear in the tree structure of the main window.

Hatch

The name of the kind of hatch used in the stratification representation, relating to the stairs and to the wall from which it corresponds. 0 means no hatch. The kind of hatch is searched for, first in the AutoCAD system files, for example acadiso.pat, and then, if there are .pat files, in the support paths.

If the hatch name starts with '-', then the hatch model is turned based on the angle rotation of the longitudinal wall axis.

Scale

The hatch scale factor.

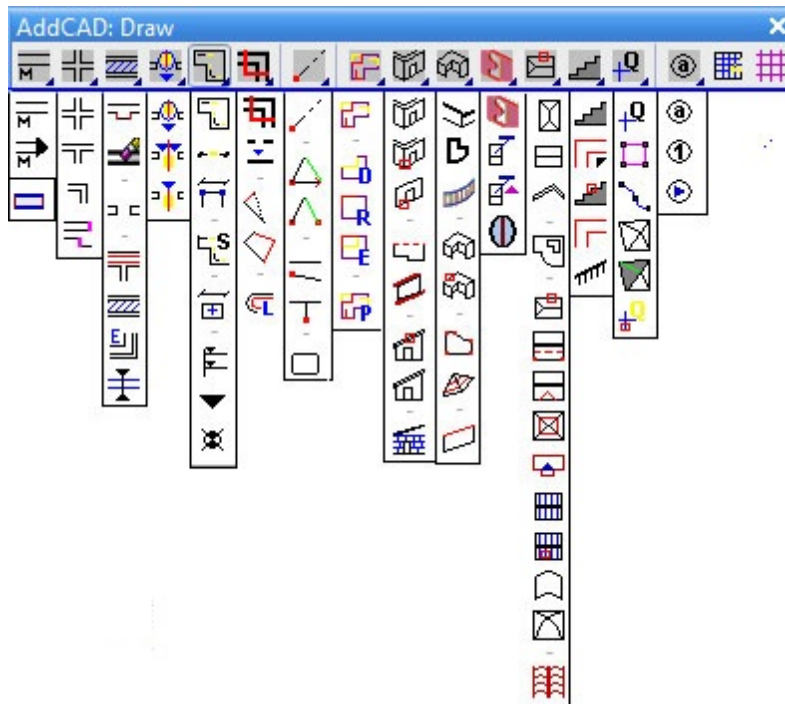
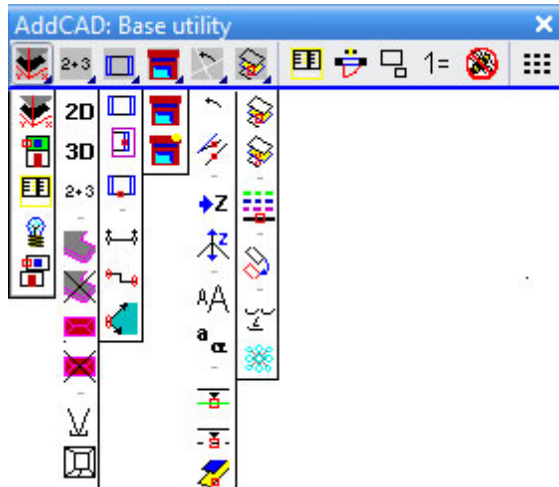
Layer

The layer name of the hatch.

Thickness

The thickness of the stratification which can be expressed in absolute dimensions in cm (10, 20, etc.) or as a composite expression of algebraic operators without parentheses. If you wish to use an expression, the following symbols can be used: \$ indicates the total thickness of the wall and s<k> indicates the thickness of the k-th stratification (k=1..n). For the stratification in the figure, we obtain a thickness equal to the total minus the thicknesses of the other four stratifications. Be careful not to define thicknesses with looped references, i.e. the second stratification should depend on the third.

Click with the left mouse button on the tool image to see the page of the command.



© © 2013 Enter your company name.
All rights reserved.

Product and company names mentioned in this manual may be trademarks or registered trademarks of their respective companies.

Mention of third-party products is for informational purposes only and constitutes neither an endorsement nor a recommendation. The author assumes no responsibility with regard to the performance or use of these products. All understandings, agreements, or warranties, if any, take place directly between the vendors and the prospective users. Every effort has been made to ensure that the information in this manual is accurate. The author is not responsible for printing or clerical errors.

The product described in this manual incorporates copyright protection technology that is protected by method claims of certain U.S. patents and other intellectual property rights.

This user manual was created with Help & Manual.